AFRL-VA-WP-TR-2006-3195

ANALYSIS AND SUPPORT INITIATIVE FOR STRUCTURAL TECHNOLOGY (ASIST)
Delivery Order 0027-03: Crack Growth and Stress Intensity Prediction Techniques: External K-Solver--Demonstration

Kyle Honeycutt

Analytical Processes/Engineered Solutions (AP/ES), Inc.
6669 Fyler Avenue
St. Louis, MO 63139

MARCH 2006

Final Report for 01 October 2002 – 31 December 2004

STINFO COPY

AIR VEHICLES DIRECTORATE
AIR FORCE MATERIEL COMMAND
AIR FORCE RESEARCH LABORATORY
WRIGHT-PATTERSON AIR FORCE BASE, OH 45433-7542

NOTICE AND SIGNATURE PAGE

*//Signature//   //Signature//

James A. Harter   Kristina Langer, Ph.D.
Program Manager   Branch Chief
Analytical Structural Mechanics Branch   Analytical Structural Mechanics Branch

//Signature//

David M. Pratt, Ph.D.
Technical Advisor
Structures Division

REPORT DOCUMENTATION PAGE		*Form Approved* *OMB No. 0704-0188*

The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. **PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS**.

1. REPORT DATE *(DD-MM-YY)* March 2006	2. REPORT TYPE Final	3. DATES COVERED *(From - To)* 10/01/2002– 12/31/2004

4. TITLE AND SUBTITLE	5a. CONTRACT NUMBER
ANALYSIS AND SUPPORT INITIATIVE FOR STRUCTURAL TECHNOLOGY (ASIST)	F33615-98-D-3210-0027-03
	5b. GRANT NUMBER
Delivery Order 0027-03: Crack Growth and Stress Intensity Prediction Techniques: External K-Solver--Demonstration	5c. PROGRAM ELEMENT NUMBER 0602201

6. AUTHOR(S)	5d. PROJECT NUMBER
Kyle Honeycutt	A02P
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER 0J

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	8. PERFORMING ORGANIZATION REPORT NUMBER
Analytical Processes/Engineered Solutions (AP/ES), Inc. 6669 Fyler Avenue St. Louis, MO 63139	

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSORING/MONITORING AGENCY ACRONYM(S) AFRL-VA-WP
Air Vehicles Directorate Air Force Research Laboratory Air Force Materiel Command Wright-Patterson Air Force Base, OH 45433-7542	11. SPONSORING/MONITORING AGENCY REPORT NUMBER(S) AFRL-VA-WP-TR-2006-3195

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES
Report contains color. PAO Case Number: AFRL/WS 06-2187, 12 Sep 2006.

14. ABSTRACT

The challenges of designing modern aircraft continue to drive the development of more advanced analytical tools; often these more advanced analytical tools themselves require development of other enabling technologies such as powerful computers and associated software. The primary historical objective of this project was to develop the infrastructure and to demonstrate that key enabling technologies such as faster and bigger personal computers, as well as database and programming software, have evolved to the point that more advanced analytical tools for analyzing the damage tolerance of aircraft structures are now possible.

15. SUBJECT TERMS
external K-Solver, stress intensity solutions, interactive stress intensity modeling

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT:	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON (Monitor)
a. REPORT Unclassified	b. ABSTRACT Unclassified	c. THIS PAGE Unclassified	SAR	126	James A. Harter 19b. TELEPHONE NUMBER *(Include Area Code)* N/A

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39-18

Table of Contents

List of Figures

List of Tables

Executive Summary

This is the final report detailing the contractual work performed in the program "Crack Growth and Stress Intensity Prediction Techniques", which builds upon the efforts of several prior contracts between APES, Anteon, and the United States Air Force.

The challenges of designing modern aircraft continue to drive the development of more advanced analytical tools; often these more advanced analytical tools themselves require development of other enabling technologies such as powerful computers and associated software. The primary objective of this project was to develop the infrastructure and to demonstrate that key enabling technologies such as faster and bigger personal computers, as well as database and programming software, have evolved to the point that more advanced analytical tools for analyzing the damage tolerance of aircraft structures are now possible.

This report describes several integration approaches used to link AFGROW crack growth analysis software with an External K-solver, finite element analysis based computational methods with crack simulation capabilities, to significantly increase the database of crack growth scenarios that are accessible to the AFGROW user community. These integration approaches include "interactive" (in which the External K-solver is called by AFGROW only when needed, and automeshing is used extensively), "table look-up" (in which the External K-solver is used to fill in a large database of Geometry Factors β and AFGROW computes Geometry Factors by interpolation and extrapolation of the database), "handbook" (similar to "interactive", AFGROW calls the External K-solver only when needed; in contrast to "interactive", the External K-solver models are developed and checked out *apriori*, and no automeshing is used), and "plug-ins" (this is a generic term describing significant capabilities added to AFGROW that facilitate communication with external components).

This project demonstrated the feasibility of the External K-solver—AFGROW integration approaches on several crack growth scenarios important to current and future aircraft designs. This program in particular focused on geometry and solutions that are important for "integral structure" applications that are being pursued for designs offering cost advantages due to manufacturing options. Many challenges were successfully overcome, thanks to evolving ancillary technologies such as computing horsepower and advances in programming and communications software. While many industry useful crack growth scenarios (including integral structures, notches, and lugs) have been added as 'plug-ins' to the AFGROW crack growth software, many more just as useful scenarios can be easily added to make AFGROW even more beneficial to the industry and to military customers. It is our estimation that the potential of this technology and the demonstrated integration approaches has only begun to be tapped by the military and the aircraft industry. The solutions provided to date offer the industry accurate advances in assessment methods that have not been available or where solution accuracy was either poor or questionable.

1 Introduction

1.1 Background

Most damage tolerance analysis tools have not kept pace with advances in computer technology in general. There continues to be a need to analyze structures with complex or unique features – for instance, the types of geometries that may be expected in integral or unitized metallic structures. To date, complex problems have been analyzed using multiple, independent analyses – usually with many simplifying assumptions built-in. This was understandable since the capability to perform more complex and detailed analysis was not available. Recent work performed by the U.S. Air Force Research Lab (AFRL) at Wright-Patterson Air Force Base, Ohio, to add a multiple crack analysis capability to AFGROW has shown that it is often not feasible to develop closed-form solutions to arbitrary geometries with more than two independent cracks. However, the capability to perform more complex analyses efficiently or even automatically is now within reach due to advances in enabling technologies such as computer software and hardware, and the time is right to move to the next level of complexity in analyses. Experience using the Microsoft COM (Component Object Model) technology has shown that this technology can allow for the integration of a third party software (called a "K-solver" in this document) that computes basic fracture mechanics parameters such as Stress Intensity Factors, which will permit real or near real time crack growth life analyses of complex geometries.

This report describes technical efforts to develop the next generation life prediction and assessment methods by seamlessly integrating crack growth and finite element method (FEM) K-solver programs. The strategy takes advantage of advances in computing technology to provide direct benefits to the USAF and the aerospace industry. The interaction of a structural FEM code that computes crack tip Mode I Stress Intensity Factors (SIFs) (sometimes referred to as a "K-solver" or "external K-solver") with a crack growth analysis code (such as AFGROW) will provide advanced life assessment and prediction techniques to the structures community. The advanced *p*-version finite element code "StressCheck$^{®}$" (ESRD, Inc., St. Louis, Missouri, USA) was used for demonstrating the methods and served as a benchmark of the required capability that would be necessary to qualify other possible K-solvers. The StressCheck$^{®}$ analysis software is quite capable of fulfilling the computational role, which demands accurate and reliable SIFs, and in addition, is able to efficiently communicate and be controlled with external programming languages via an Application Programming Interface (API), specifically, Microsoft's industry-standard COM. Because the USAF/AFRL crack growth code AFGROW is itself one of the few crack growth codes offering an API (also specifically, Microsoft's COM), StressCheck$^{®}$ and AFGROW are jointly very well-suited for interactive programming.

Several demonstration cases that are typical examples of the integration of AFGROW with a K-solver have been provided in Chapter 2. A discussion of the actual Deployment of the External K-solver and the advances in AFGROW technology that were necessary is found in Chapter 3. Conclusions and Recommendations for future work are found in Chapter 4.

1.1.1 Previous Work

Recent experience documented in the final report for a previous U.S. AFRL contract (F33615-98-D-3210 Delivery Order No. 27-3 S1126), Reference [1], has shown that key enabling

technologies which will allow the efficient integration of a third party K-solver with widely used crack growth analysis software are now readily available, thereby permitting near real time or real time life analysis of complex geometries. These technologies include computer hardware that is much more capable (faster speeds, more memory) and affordable than ever before, as well computer software such as the widely-available Microsoft COM (Component Object Model). Many developers' third party K-solvers have used these technologies to improve efficiency and access to their capabilities. One such K-solver that uses the *p*-version of the finite element code, StressCheck® (ESRD, Inc.) has been used to demonstrate integration of a K-solver with the crack growth software AFGROW. ESRD has added COM technology to their code to facilitate integration with AFGROW and has worked with quite closely with Analytical Processes/Engineered Solutions (AP/ES) to make this technology work.

The previous project made use of the expertise of individuals who possessed a breadth of knowledge and experience in the enabling technologies. State-of-the-art computational capability for an external K-solver and projections of future computational techniques were obtained from the engineering expertise of ESRD, Inc., the developers of StressCheck®. Invaluable descriptions of the internal structure of AFGROW, experience with the user community, and computational needs were provided by Jim Harter of AFRL/VASM and his support personnel. Industry analysts specializing in commercial applications, military transport, fighter, and engine damage tolerance assessments were consulted; along with experts in databases and advanced programming. The experience base also includes developers of fatigue and fracture mechanics methods, procedures, processes, requirements and criteria.

The primary objective of this earlier project was to describe the infrastructure and guidelines to evolve the technology, to plan and prioritize activities, and to ensure the USAF and industry are able to capture the benefits of this technology. The final report described a suggested funding profile that can impact aircraft structural integrity in the near future. The virtually infinite number of structural geometries, loading and cracking configurations were classified into a few Problem Classes. To aid in setting priorities and evaluating the level of technical skill needed to construct an integrated solution, a Complexity Rating Matrix was presented. For each Problem Class, the need for the specific problem solution was documented and an estimate of the Problem Class's complexity and relative priority within the aerospace industry was provided by using a substantiation method which categorically addressed the technical, business and integration cases for the Problem Class. A Solution Strategy or Approach was proposed for each problem class. Due to advances in software technology such as Microsoft COM (Component Object Model), it was estimated that Interactive Solution Strategies are presently viable for several Problem Classes, while for other Problem Classes, the current software technology is still not up to speed and would require solutions that would be unwieldy except where often-used internal solutions could be built, and so a Table Look-up Strategy would be more appropriate. For many relatively simple structural geometry and load conditions, a Handbook Strategy would work very well. Finally, for some of the more complex Problem Classes, integrated solutions are simply not feasible or practical at all, at least in the near term.

A system specification with enough detail to allow the integration with AFGROW of any K-solver was described. Requirements for the end integrator of the K-solver and AFGROW product; i.e., what the integrator must be capable of, what criteria must be defined, etc. was also defined. Finally, several useful examples of the integration of AFGROW with a K-solver, in this case ESRD, Inc.'s StressCheck®, were described.

Four integration approaches or "Solution Strategies" that could be exploited immediately were identified in the previous project: Interactive, Plug-in, Handbook and Table Look-up. The characteristics (complexity, etc.) of the problems themselves determine the Solution Strategy used. For instance, the complexity of the analysis of a crack growing in a large integral or unitized structure appeared to be high; however, due to the symmetry and relative simplicity of the geometry and loading, this analysis could be handled by the combination of an Interactive and a Table Look-up by using auto-meshing for cracks of almost all lengths, while using Table Look-up wherever gaps occurred. The four Solution Strategies and the Problem Cases chosen to demonstrate the feasibility of the approaches are shown below:

<u>**Table 1**. Solution Strategies and Problem Cases Summarized</u>

Solution Strategy 1: Interactive (3 Problem Cases)

Continuing damage in an integrally stiffened panel

Two bay crack in integrally stiffened panel

Stiffener adjacent to cutout in skin (replaced with MSD model)

Solution Strategy 2: Plug-in (1 Problem Case)

Crack growth rate

Solution Strategy 3: Handbook (1 Problem Case)

Variable notches

Solution Strategy 4: Table Look-up and Multiple Cracks (up to 6 Problem Cases)

Lug Part-through Crack (PTC)

1.1.2   2004 Project Approach

Feasibility and applicability to actual aircraft industry problems will be demonstrated for the four Solution Strategies identified in the previous section: Interactive, Plug-in, Handbook and Table Look-up, by integration of the External K-solver StressCheck® with AFGROW. StressCheck® is a commercially available *p*-version finite element method software code created by ESRD, Inc., St. Louis, MO, USA. Two- and three-dimensional finite element models with various crack scenarios will be assembled into model libraries that will be available on a public access server, thereby eliminating the need for the user to have her own license for StressCheck®. The finite element models will be fully-integrated into AFGROW--user interfaces were designed to look as close to AFGROW as possible, to significantly shorten the initial familiarization period associated with any new software or upgrade; if the user is familiar with AFGROW, they should have no problem running and obtaining good results from the External K-solver—AFGROW integrated software. The finite element models that were built in this project were identified and prioritized by participants from the Boeing Corporation, thus bringing the Original Equipment Manufacturer's (OEM's) perspective of applications that can use this advanced technology of

interactive life assessment methods. To accomplish the goal of providing these solutions and tools to assist the user community, the following considerations were addressed: 1) solutions would be of a sufficient range to cover many real-world cases, while also being of manageable size to fit the program, 2) solutions were to have demonstrated accuracy, while also being generated in reasonable CPU run times, 3) robustness of the solutions was to be determined and solution limitations clearly defined, and 4) rules and guidelines for using the new models were to be incorporated into the user analysis procedures.

1.2 Participants

The primary customer for this contract is the U.S. Air Force Research Laboratories at Wright-Patterson Air Force base in Dayton, Ohio, who are the developers of the crack growth analysis computer software AFGROW. The work performed by APES under this effort is directly applicable to enhancing that AFRL software, as well as advancing the state-of-the-art for crack growth and damage tolerance predictions in the aerospace and other commercial industries. The Anteon Corporation serves as the direct contractor to the U.S.A.F., providing the experience and contractual expertise required for dealing with the U.S.A.F. APES, Incorporated serves as the primary technical lead on this contract. LexTech, Inc. provides the ability to modify and release the AFGROW code in support of the contractual goals. The Boeing Company provides technical advice, a link to aircraft field problems, some components of the technical effort, and arranges the mechanical test effort. Purdue University provides the mechanical test support for this program through Boeing.

2   Technology Development

2.1   Overview

The technology development required to implement the four Solution Strategies: Interactive, Plug-in, Handbook and Table Look-up is summarized in this Chapter. The four Solution Strategies and the Problem Cases chosen to demonstrate the feasibility of the solution approaches were shown in Table 1.

Each Solution Strategy and Problem Case will be described in terms of geometry, loads, boundary constraints, and solution strategies. Challenges, limitations and recommendations for future study (either to enhance the Problem Case or to continue to expand the models that can be accessed within each Solution Strategy) will be described before moving on to the next Solution Strategy.

2.1.1   Selection of External K-solver code

The AFGROW integration solution for Each Solution Strategy and Problem Case will be constructed using the same External K-solver, StressCheck®, which is a sophisticated *p*-version finite element method software created and supported by ESRD, Inc. (St. Louis, MO). StressCheck® was chosen as the External K-solver because:

- It contains an efficient and accurate Contour Integral Method (CIM) algorithm to calculate reliable Stress Intensity Factors (SIFs) for corner and surface cracks in arbitrary structural geometry, Reference [2],

- Its excellent error checking procedures (primarily through its capability for automatic *p*-extension and numerical convergence checks) will begin to address the shortcomings of well-known and frequently used 'historic' SIF solutions,

- It is one of the few commercially available *p*-version finite element method software codes that use an engineering Handbook interface, Reference [3]- StressCheck® User Manual(s), which facilitates and expedites parametric variations of geometry, load and crack configuration, and

- It is one of the few commercially available FEM codes that allow user interaction through the Microsoft Component Object Module (COM) programmatic interface.

2.1.2   Model Convergence Issues

StressCheck® uses some of the most sophisticated and robust error checking procedures to guarantee accuracy and reliability to user pre-solution standards. Numerical convergence was obtained by *p*-extension in which the polynomial level of the approximation functions was increased from 6 to 8, and by *h*-extension in which the finite elements are appropriately refined near areas of large stress gradients; therefore the solutions are *hp*-convergent, the best available numerical technique. These procedures include checking of global convergence by monitoring convergence of the error in the energy norm, and checking of the local convergence of the Stress Intensity Factors along the entire crack front. All StressCheck® computed SIFs were checked and verified to be numerically converged to within 2%, with the overwhelming majority of the SIFs numerically converged to within 1%.

2.2 AFGROW External K-Solver: Interactive Solutions

2.2.1 Objective

The Interactive Solution Strategy will be demonstrated by fully integrating the External K-solver with AFGROW software for three-dimensional integral or unitized structures. AFGROW will call the External K-solver only when it needs Stress Intensity Factors (SIFs) for its crack growth calculations. Crack dimensions will be updated in the External K-solver and updated SIFs will be passed to AFGROW as the crack growth calculations continue. The AFGROW-External K-solver interaction continues until AFGROW (through Preferences input by the user) decides to stop the crack growth; crack growth can stop due to K_{max} reached, a specified maximum crack length, a specified number of cycles, and a specified user-defined K_{max}, among other criteria.

2.2.2 Description

To demonstrate the Interactive Solution integration, integral (unitized) structural configurations were chosen after cooperative discussions among all project partners. These structures were chosen because of their importance in current and future aircraft, and solutions for typical examples of these structures are obtainable in the near term. Integral structural models can be used to simulate many different substructures in an airplane: upper and lower wing skins, bulkheads, keel beams, ribs, webs to flanges, skins to stiffeners, etc. The models that were chosen can be parameterized for a wide range of geometry variations, including panel thickness, blade or stiffener thickness, and variations to spacing and blade types.

ESRD, Inc. in an earlier project (some details discussed in Reference [1]) demonstrated a limited case for a wing application for Boeing's "Unitized Structure Program" that illustrated some of the concepts of obtaining K solutions for unitized structure, but the demonstration did not include tracking crack propagation that typically occurs in a crack growth code. At the time of the ESRD implementation, Boeing expressed interest in the continued pursuit of this type solution for integral wing panels, and particular C-17 'big boned' geometries were also identified as a candidate for improving affordability of the C-17. The Interactive procedure allowed formulation and checking of integral panel solution features such as crack arrestment and shape penetration into stiffeners. Sensitivity studies provided insights into merging the technical capability with a business case and defining the solution integration plan. Some moderate proof-of-concept element tests and exploratory building block test programs were performed to substantiate the technology, and to assist in developing demonstration/validation (DEM/VAL) plan for future testing under existing or future aircraft programs.

The Interactive Solution Strategy was not without challenges. For example, an interactive integration may require dealing with one or more crack fronts and/or directions – each advancing under its own separate da/dN rates and possibly transitioning separately from through-cracks into elliptical cracks or vice-versa at geometrical junctions. Local stiffener failures or stable crack progression may continue past the stiffener juncture and emerge as two advancing crack fronts, one in the stiffener and one in the panel again, thus potentially 3 or 4 crack fronts require an infrastructure that can handle simultaneous propagation of multiple cracks. The interacting codes (AFGROW and External K-solver) must keep track of the changing crack configurations, monitoring switches from single to multiple and back to single cracks, with at least two distinct crack fronts being controlled separately.

Two compatible implementation architectures were considered for this Solution Strategy: a basic Microsoft Visual Basic ActiveX Dynamic Link Library (DLL)—Component Object Module (COM) architecture and a second, internet-accessible single computer server. In the first architecture, a

separate DLL is compiled and can be distributed with the AFGROW executable. This DLL would require that the user's machine have a licensed copy of the basic K-Solver software and any additional licensed modules if applicable (for example, automeshing or fracture mechanics modules). The second architecture uses the same code architecture, but distributes the software on a single computer server with licensed native versions of the External K-solver software , AFGROW, the controlling DLL, and the K-Solver software. User access and licensing uses Windows Terminal Services over the Internet, and a 'rental' or 'royalty' fee can be used to satisfy External K-solver licensing requirements. This second option has the potential for providing user access to integral structure crack growth solutions at a much lower startup cost than the first option would allow, and also allows better developer control over how the software codes interact when versions are updated. However, there are also disadvantages inherent to the second approach, chief among them is that corporate firewalls may limit access to the server via Windows Terminal Services and that the multi-threading capability (i.e. for simultaneous users) of AFGROW and the DLL's is questionable.

The second architecture was used to demonstrate feasibility of the Interactive Solution approach, primarily due to its benefit during periods of rapid software development and frequent software releases or beta releases. A native AFGROW version is integrated with the DLL's and the External K-solver on a internet-accessible single server that can be accessed by using the Remote Desktop Connection through Windows Terminal Services. Geometry and loads are controlled by the user through the AFGROW GUI. AFGROW communicates with individual DLL files, which in turn communicate with the External K-solver via Microsoft's COM technology. The COM-enabled server integration includes all functions and methods necessary for setting and retrieving geometric, cracking, loading, and constraint properties, as well as returning Stress Intensity Factor, numerical convergence information, and error messages. The COM server is also responsible for selecting the appropriate StressCheck® models from the mesh library that provides the best convergence for the current cracked state and geometric parameters. APES contracted with LexTech, Inc. to perform all stages of integrating the COM DLL's into the AFGROW code. The current AFGROW framework is used for setting crack propagation limits, defining output requirements, and visualizing the crack progression.

2.2.3   Integral Structure - Continuing Damage

A representative crack growth scenario in a large integral or unitized structure was modeled by integration of AFGROW with the External K-solver. The structure is a large flat plate with an integral stiffener and a small hole located close to one edge of the plate, loaded with a uniform tension at far field. The Crack Scenario Nomenclature is described by Figure 1 below:
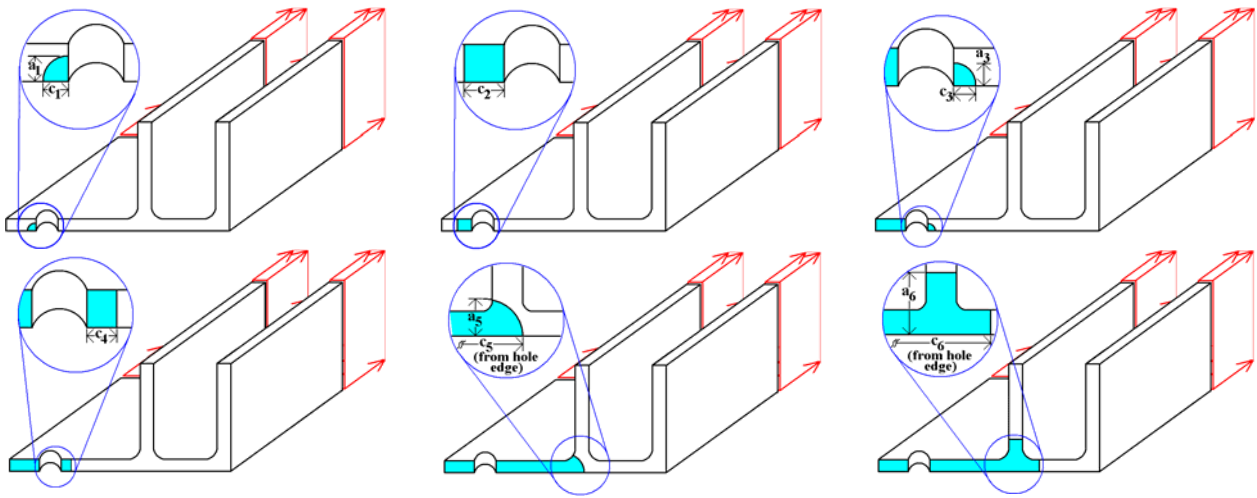
Figure 1 Crack Scenario Nomenclature for Crack Growth in a Tension Loaded Plate with Integral Stiffener and Single Offset Hole.

The postulated crack growth scenario begins with a small elliptical corner crack on the side of the hole closest to the plate edge (Crack Stage 1), which transitions to a through crack growing toward the plate edge (Crack Stage 2). After the through crack breaks through to the edge, the crack growth continues on the hole side opposite to the plate edge with a 0.005 inch elliptical corner crack at the hole (Crack Stage 3). The corner crack is allowed to grow until the crack transitions to a straight-through crack (Crack Stage 4), which continues to grow toward the upright stiffener. As the crack enters the fillet of the stiffener closest to the hole, the crack transitions back to an elliptically shaped part-through crack (Crack Stage 5). The crack continues to grow in elliptical shape until the crack front reaches the fillet on the side of the stiffener opposite to the hole, where it breaks into two independently growing straight cracks, one up the stiffener, the other continuing in the plate away from the stiffener (Crack Stage 6). Each crack grows independently until one of the user-specified failure criteria is met.

Project partners, especially the Boeing Corporation, hypothesized that the Crack Stages 1 and 2 could be handled sufficiently with current methods; therefore the crack stages that were modeled were the remaining Stages 3 to 6; this is the so-called 'Continuing Damage' crack growth scenario, Figure 2. To implement, the External K-solver was used in two ways: a full spectrum of models needed to cover a wide range of elliptical corner crack shapes and dimensions (Crack Stage 3) was constructed; the models are used interactively, being called by AFGROW only when new SIFs are needed. After the elliptical corner crack breaks through to the plate back face, the crack transitions to a straight through crack (Crack Stage 4). Crack growth from Crack Stage 4 through Crack Stages 5 and 6 to panel failure is handled by the External K-solver with an automatic meshing (AKA "automeshing") procedure developed under the direction of ESRD; this automeshing procedure is

also used interactively, being called only when needed by AFGROW. The logic that tells AFGROW when to call the External K-solver interactively, and when to use the automeshing procedure are implemented through COM calls.
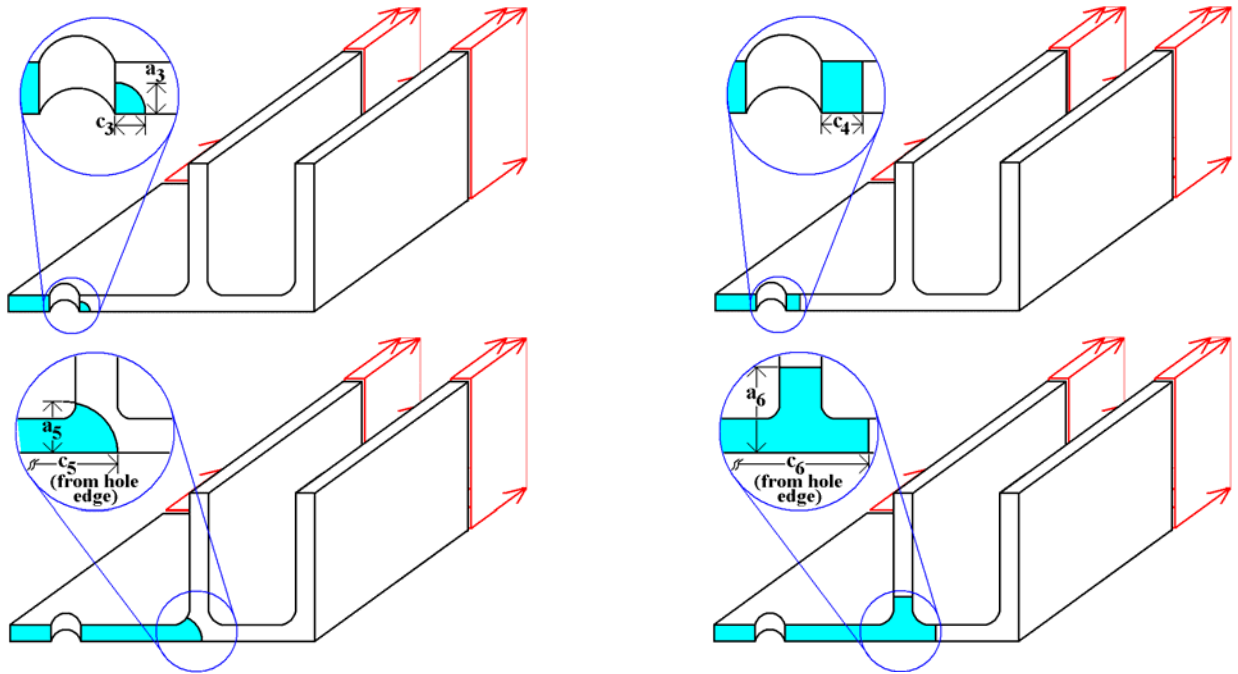
Figure 2. Crack Scenarios That Can Be Analyzed with the Interactive Solution Procedure

The integral structure is shown with the parameters that can be controlled by the AFGROW user in Figure 3. The Table 2 contains the list of parameters and the restrictions on the range. Often, parameters in the models can be adjusted to dimensions outside the range indicated in the Table; however, if the analyst chooses to do so, numerical accuracy in the finite element models might exceed our imposed criteria and the solution suffer unacceptably.
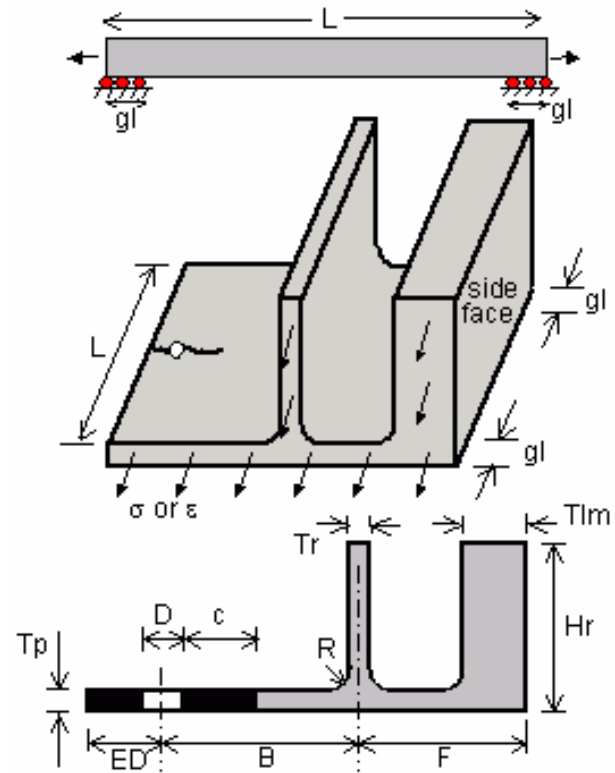
Figure 3. Integral Structure—Continuing damage problem parameters, shown in side view, trimetric view, and view of cracked cross-section

Table 2. Range of Parameters for Integral Structure—Continuing Damage Model.

Parameter	Description	Range
a	Crack Bore Length	$0.005 \leq \mathbf{a} \leq 0.9\mathbf{Tp}$
B	Distance of Stiffener from Hole	$3\mathbf{Tp}+0.5\mathbf{D}+\mathbf{R} \leq \mathbf{B} \leq 10$
c	Crack Surface Length	$0.5\mathbf{a} \leq \mathbf{c} \leq 2\mathbf{a}$
D	Hole Diameter	$0.1875 \leq \mathbf{D} \leq 0.5$
ED	Distance of Hole from Near Edge	$1.5\mathbf{D} \leq \mathbf{ED} \leq \mathbf{B}$
F	Distance of Stiffener from Edge Away from Hole	$0.5\mathbf{Tr}+2\mathbf{R}+\mathbf{tlm}+0.03 \leq \mathbf{F} \leq$ $0.5\mathbf{Tr}+2\mathbf{R}+\mathbf{tlm}+20$
gl	Grip Length	$0 \leq \mathbf{D} \leq \mathbf{L}/2$
hlm	Lumped Mass Height	$\mathbf{Tp}+\mathbf{R}+0.03 \leq \mathbf{hlm} \leq 2\mathbf{Hr}$
Hr	Stiffener Height	$\mathbf{Tp}+\mathbf{R}+0.25 \leq \mathbf{Hr} \leq \mathbf{Tp}+\mathbf{R}+5$
L	Panel Semi-Length	$\mathbf{B} \leq \mathbf{L} \leq 4(\mathbf{ED}+\mathbf{B}+\mathbf{F})$
Modulus	Young's modulus	No limit
Nu	Poisson ratio	$0 \leq \mathbf{Nu} < 0.5$
R	Fillet Radius	$0.125 \leq \mathbf{R} \leq 0.5$
Tlm	Lumped Mass Thickness	$\mathbf{Tr} \leq \mathbf{Tlm} \leq \text{fcn}(\mathbf{F})$
Tp	Panel Thickness	$0.1 \leq \mathbf{Tp} \leq 0.5$
Tr	Blade Thickness	$0.1 \leq \mathbf{Tr} \leq 0.5$

The user can specify a constant stress on the two far-field boundaries. In addition, to model more closely the experimental set-up or the rotational edge capability of aircraft integral structure, the application of test fixture grips or multiple rows of attachment fasteners on the ends of the structure is allowed. This constraint is modeled with a fixed normal displacement over a specified grip length (**gl**), on the bottom of the plate near each far-field boundary, away from the crack face. The actual length of the experimental integral structure can also be controlled by the user. Some representative values of Stress Intensity Factors given particular combinations of plate length and grip length are given in Table 3. It is clear that the plate length has a much bigger influence on the SIFs than the grip length.

Table 3. Effect of Variation of Plate and Grip Lengths on SIFs in Integral Structure.

Plate Length, in.	Grip Length, in.	ΔK, ksi-in$^{1/2}$	Δ %
22.2	1.5	0.4364	0
10	1.5	0.4697	+7.63
5	1.5	0.7278	+66.77
22.2	5	0.4258	-2.66
22.2	10	0.4087	-6.35

2.2.3.1 Demonstration Code (pre-AFGROW integration)—Cracked Stiffened Panel

Before modifications to the AFGROW software were made, External K-solver integration software was constructed to demonstrate the feasibility of the Interactive Solution Strategy; this software was demonstrated by APES at seminars at Aging Aircraft 2003 in New Orleans, LA, and at The U.S. Air Force Aircraft Structural Integrity Conference 2003 in Savannah, GA. A prototype Windows executable application communicated with StressCheck V6.2 through COM to automatically construct the finite element mesh for a cracked stiffened panel given its dimensions and crack length. The application created a 2D profile of the stiffened panel, extruded the profile into a solid body, automatically meshed the front face of the solid, extruded the resulting 2D mesh into a specified number of layers of 3D elements, and then inserted local mesh refinements at both leading and trailing the crack front(s). Materials and boundary conditions were assigned and the model was solved using *p*-extension. Stress Intensity Factors (SIFs) are computed and extracted at a set number of points along the crack front(s). These SIFs were used by the application to perform incremental crack growth. AFGROW was used by the custom application to provide crack growth (da/dN) rates, given a delta-K value. The cracked stiffened panel was a developmental model of the Integral Structure—Continuing Damage model described in the previous section, Figure 4. Similar to the Integral Structure models, the Crack Stiffened Panel demonstration model classified cracking scenarios in terms of 6 stages—the first two stages were not analyzed with the demonstration code. Stages 3 to 6 represented cracking from a part-through crack at a single offset (unloaded) hole (Stage 3), transitioning to a straight crack that is 'through' the panel, between the hole and the central upright integral stiffener (Stage 4), propagating was an elliptical through crack through the fillet (Stage 5), breaking into two straight through cracks (Stage 6) until the structure fails, Figure 5. Parameter definitions and nomenclature are exactly the same as in the Integral Structure—Continuing Damage models.

This 'cracked stiffened panel' model is the same model domain and has almost the same loads and boundary conditions analyzed in the Integral Structure—Continuing Damage model. However, in contrast to the approach used in the Integral Structure—Continuing Damage model, which was accessed directly through the AFGROW interface, this demonstration model was accessed and solved through the customized Visual Basic application. A Microsoft Excel spreadsheet was used for user input and output. This architecture was a necessity since a demonstration of the technology was an early requirement of the program, and changes to AFGROW to accommodate custom models

could not be made in this required timeframe.

To facilitate implementation of the demonstration code, there were several crack propagation assumptions or 'rules' established for this demonstration code:

- Through Cracks: The single Stress Intensity Factor used in the crack propagation is the average of 5 Stress Intensity Factors computed across the entire crack front.

- Transition from Stage 4 to Stage 5: Use Initial "**lc**" and "**a**" inputs

- Part-Through Cracks, Stage 5: Initial "**lc**" value assumed, ellipse center point remains fixed (**lc** and **c** increments are equal)

One problematic issue was establishing the rule by which a through crack that is elliptically shaped would propagate—at issue is the method used to compute the new crack dimensions given the SIFs along the crack front when the crack front intersects the left (or forward) fillet (Stage 5). The method chosen for the demonstration calculated the average of the SIF on each of two halves of the elliptical crack front—the crack increment Δc is along the bottom of the panel (side opposite the stiffeners) is computed from the SIF averaged over the half of the crack front closest to the bottom; the crack increment $\Delta a'$ is computed by averaging the SIFs along the top half of the crack front. Then, Δa is calculated from the following equation:

$$\Delta a = \Delta a' \sin(\phi) \qquad (1)$$

in which ϕ is the elliptical angle that is defined by the intersection of the circular fillet edge and the elliptical crack front.

- Transition from Stage 5 to Stage 6: Occurs when leading crack mesh touches right fillet. Instantly switch to two through cracks 0.02" outside fillet tangencies.
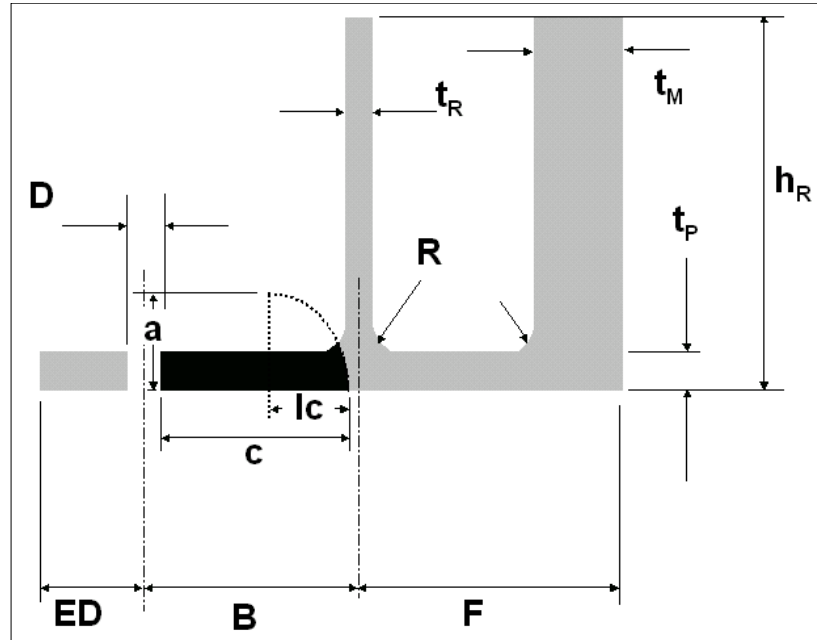
Figure 4. Crack Stiffened Panel Analyzed in Demonstration Code. Uniform Tension Can Be Applied at Far Field. Nomenclature and Parameter Definitions Identical to Integral Structure Continuing Damage Model.
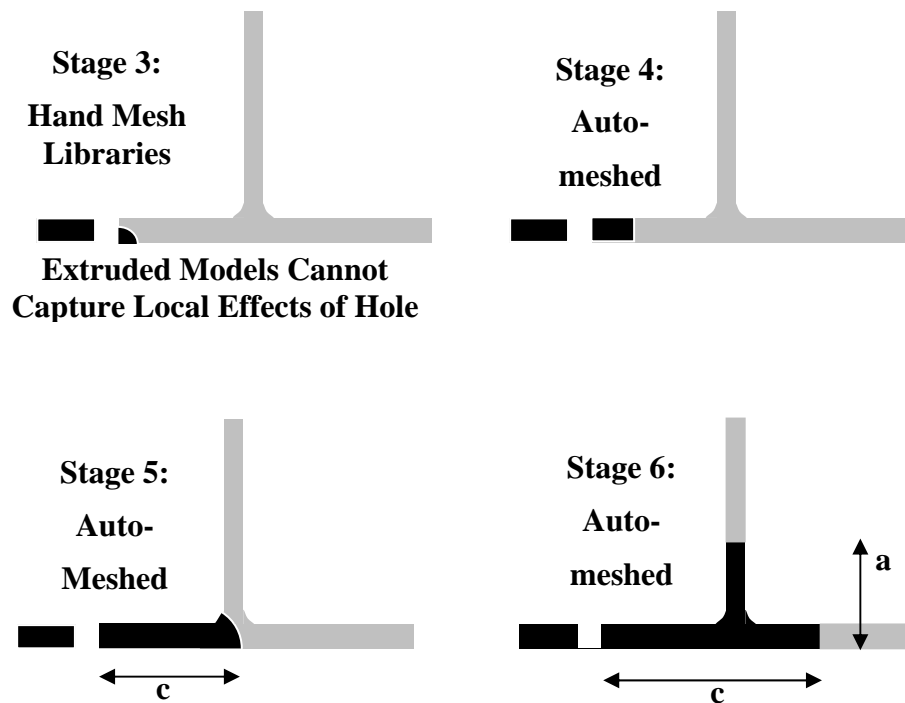
Figure 5. Five (5) Crack Stages Delineate Cracking Scenarios Through Failure (Lumped Masses Have Been Removed from the Figures).

2.2.3.2 Mechanical Test plan

In a separate but related program, Purdue University is testing a number of integral structures that are constructed entirely of PMMA (polymethyl methacrylate), a type of clear plastic, to facilitate crack propagation tracking. Purdue planned to test two structural types; Stiffened Panel (Conventional Design) and Continuing Damage Crack from Hole. One major goal of these tests was to successfully capture the crack shapes and crack tip Mode I Stress Intensity Factors (SIFs), ΔK. The crack shapes can be measured optically during the test, and SIFs are determined by estimating the crack growth rates da/dN, extracting the ΔK from the baseline material crack growth rate curves. Preliminary experimental details and comparisons will be summarized here.

The Stiffened Panel (Conventional Design) is a large flat plate with two large upright stiffeners which are integral to the plate, that is, they have been machined from the same block of material as the plate, and therefore do not need to be fastened. One of the stiffeners is 'thin' and roughly down the center of the plate, while the other stiffener is relatively large "lumped mass"—its purpose is to simulate additional stiffeners in an aircraft component and reduce the bending stress in the plate. There are three (3) configurations that will be tested, Figure 6—the only differences in the three configurations are the thickness of the 'thin' central stiffener and the overall width of the panel. Physical dimensions of each of the three configurations can be found in Table 4. The applied stress ratio is $R = 0.3$, and the applied stress is 0.150 ksi.

Table 4. Stiffened Panel (Conventional Design) Test Matrix.

Configuration	'Thin' Stiffener Thickness, in.	Total Width, in.	Large Stiffener Thickness, in.	Skin Thickness, in.	Number of specimens
I	0.2	7.26	1.0	0.25	3
II	0.4	7.46	1.0	0.25	3
III	0.6	7.66	1.0	0.25	3
				Total:	**9**
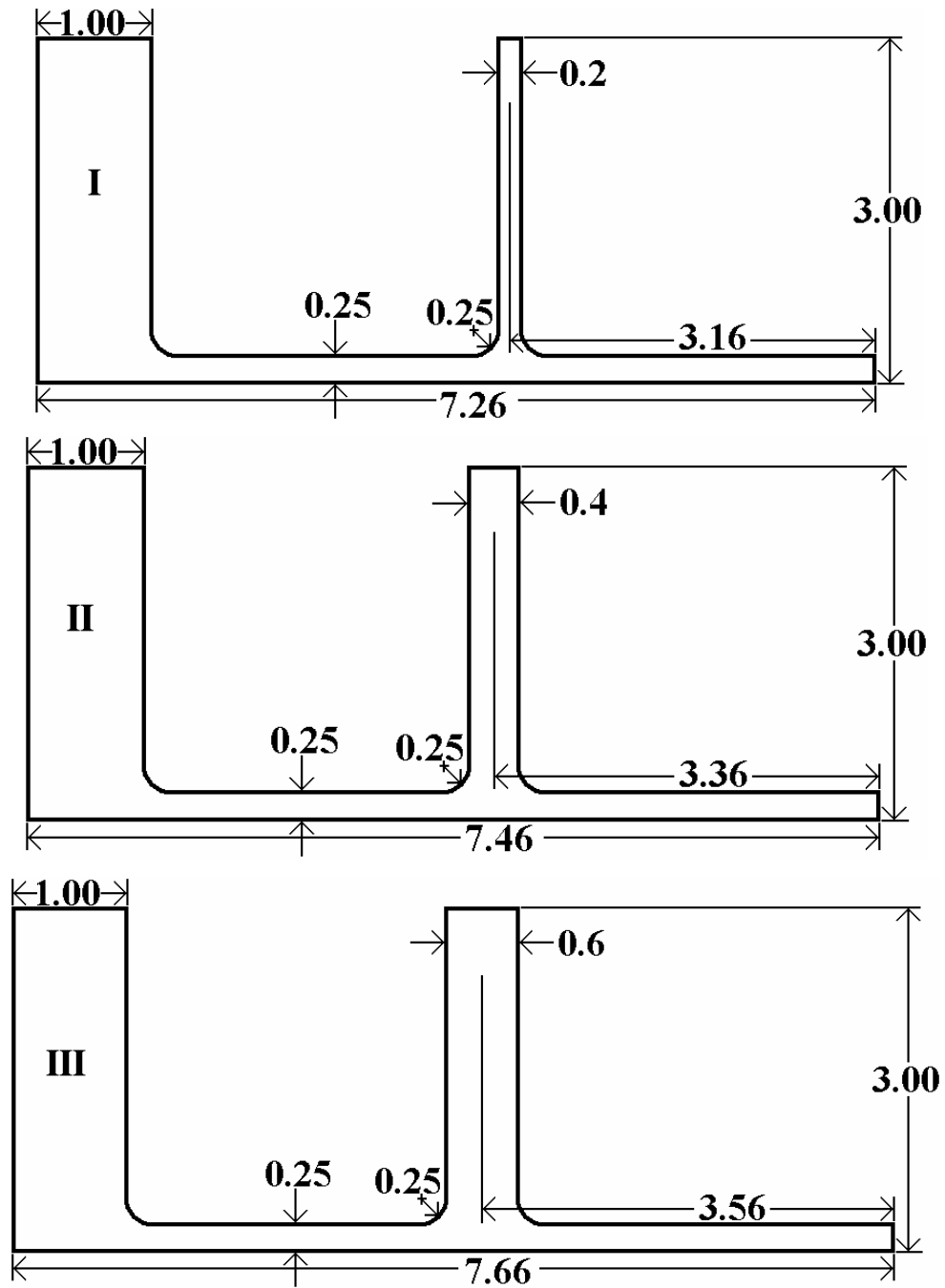
Figure 6. Three Stiffened Panel Configurations. Crack Growth is from Right to Left in these Figures.

The Continuing Damage Crack from Hole structure simulates crack propagation in a large flat plate that originally was attached to a much large structure, like the frame in a fuselage with an offset hole or some other attached structure. Cracks often nucleate on the side of the hole closest to the attach structure (the right side of the holes in Figure 7), propagate until the ligament closest to the attach structure fails, then crack propagation picks up on the hole side opposite to the attach. The tested structure simulates crack propagation after the ligament has broken through and cracking continues on the hole side opposite to the attach structure (the left side), Figure 7. Table 5 summarizes many of the physical dimensions. The length (into the page) of each plate in Figure 7 is is 16 inches.
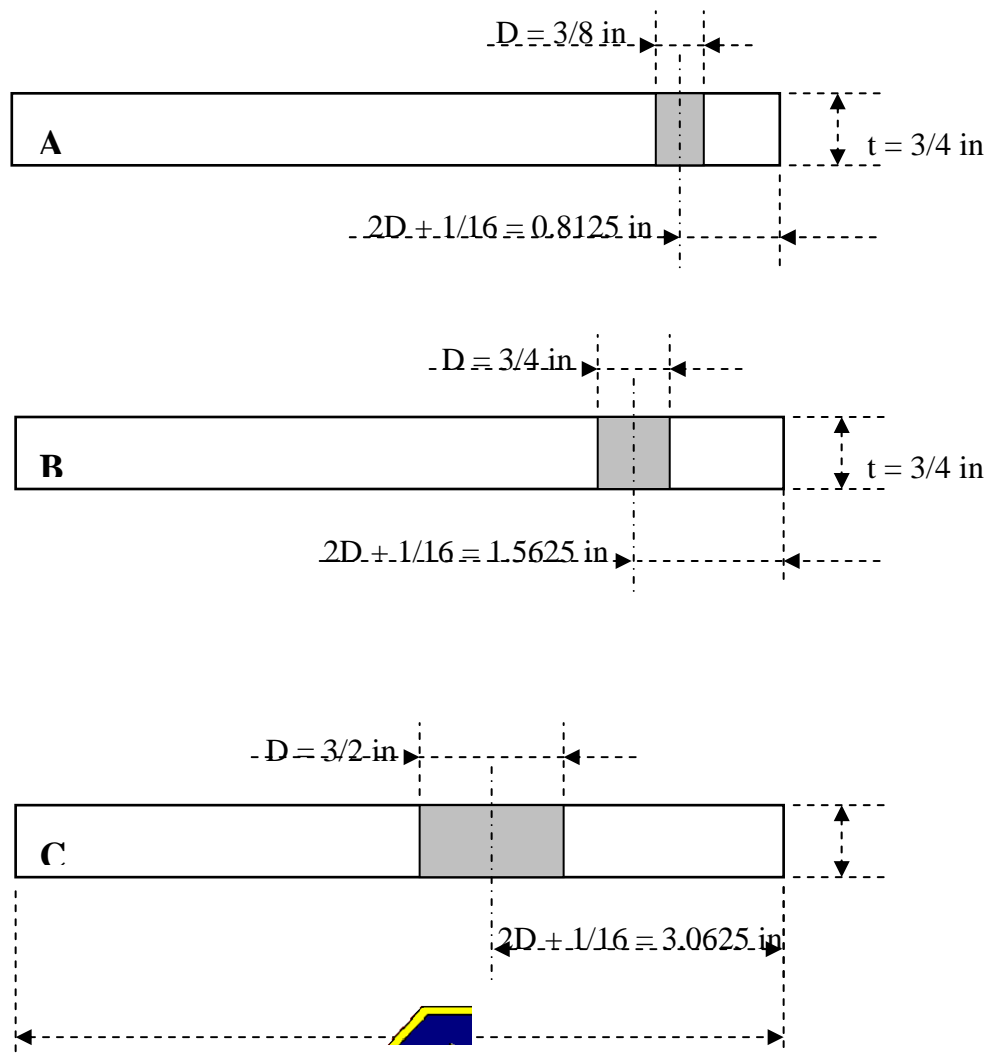
Figure 7. Three Continuing Damage Crack from Hole Configurations.

18

Table 5. Continuing Damage Crack from Hole Matrix.

Configuration	Hole Diameter (D), in.	Plate Thickness (t), in.	D/t	2D + 1/16, in.	Number of specimens
A	0.375	0.75	0.5	0.8125	2
B	0.750	0.75	1.0	1.5625	2
C	1.500	0.75	2.0	3.0625	2
				Total:	**6**

2.2.3.3 Mechanical Test Results

Sample test results collected by Purdue University in a separate but related program whose test plan was discussed in the previous section are summarized here for a single configuration (Configuration I) of the integral structure component. Two fundamental types of data were measured: crack shapes (or modes) and Mode I Stress Intensity Factors (SIFs). The PMMA facilitates the description of the crack shapes, Figure 8, where crack propagation and development is described for Configuration I. Figure 9 shows ellipses fit to the mode shapes—these fits will be used later to determine crack dimensions in finite element analyses described in later sections.

2.2.3.4 Mechanical Test Correlations

Future work related to this contract will involve a complete evaluation and correlation of the integrated External K-solver AFGROW predicted crack growth with the Purdue experimental results. The results of all of the tests were not available to the authors of this report by December 2004, and are not included in this report.

For completeness, this report will describe preliminary correlation with early Purdue test results for Configuration I. The eyeball-fit ellipses in Figure 9 were used as input parameters in the finite element analyses. Computed SIFs were extracted from the integrated AFGROW-External K-solver interface and compared to the experimentally estimated SIFs, Figure 10. The SIFs were estimated from the experimental crack growth increments and knowledge of the cycle counts—crack increments divided by number of cycles for a small cycle count yield an average crack growth rate, da/dN. This value of da/dN was used with the baseline material crack growth rate curve to extract the ΔK for a given stress ratio (in this case, for $R = 0.3$). For straight crack fronts, the average SIF across the crack front is reported at the average crack length as measured in the experiment. For elliptical crack fronts, the SIF for the *'c'* dimension (along the bottom surface) is the average of the SIF's on the lower half of the crack front, and the SIF for the *'a'* dimension is the average of the SIF's on the upper half of the crack front. On Figure 10, the percentage difference between the computed SIFs relative to the experimental SIFs are shown. These percentage differences were similar to those found in the comparisons of the SIFs for the *'a'* crack dimension (not shown).
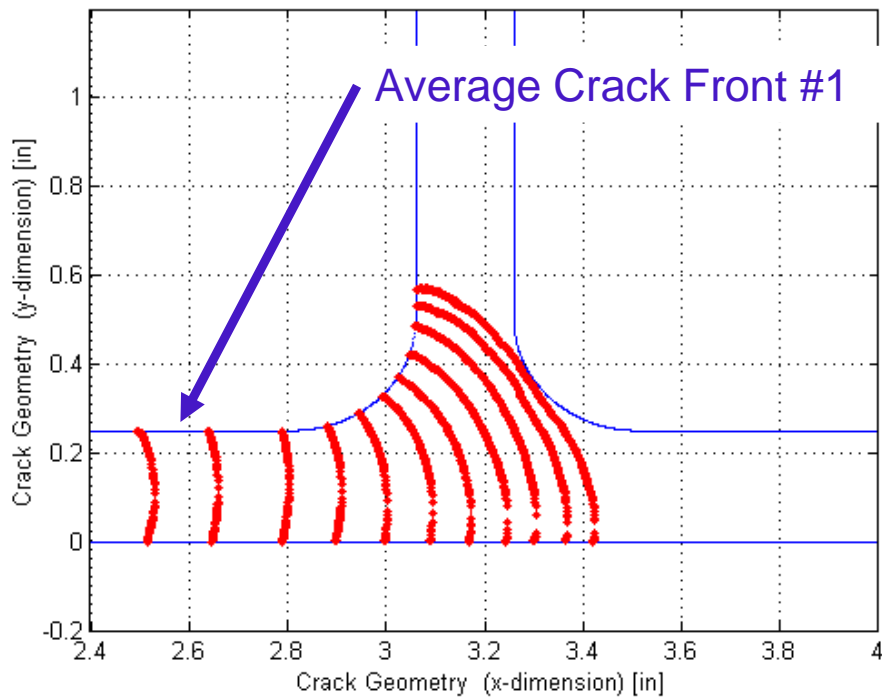
Figure 8. Crack Shapes in Stiffened Panel (Conventional Design) Configuration I.
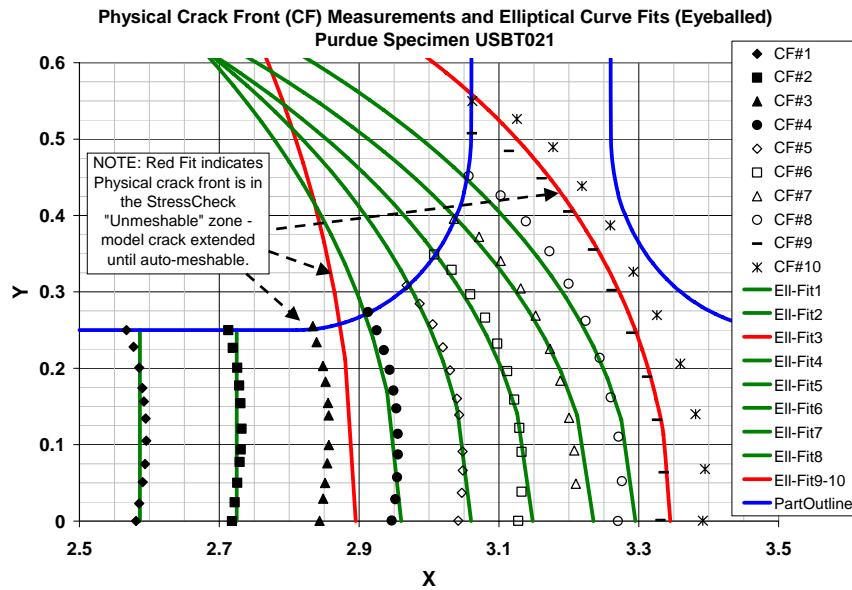
Figure 9. Crack Shapes Fit to Ellipses in Stiffened Panel (Conventional Design) Configuration I. Fit Ellipses Used to Define Crack Dimensions in Finite Element Analyses.
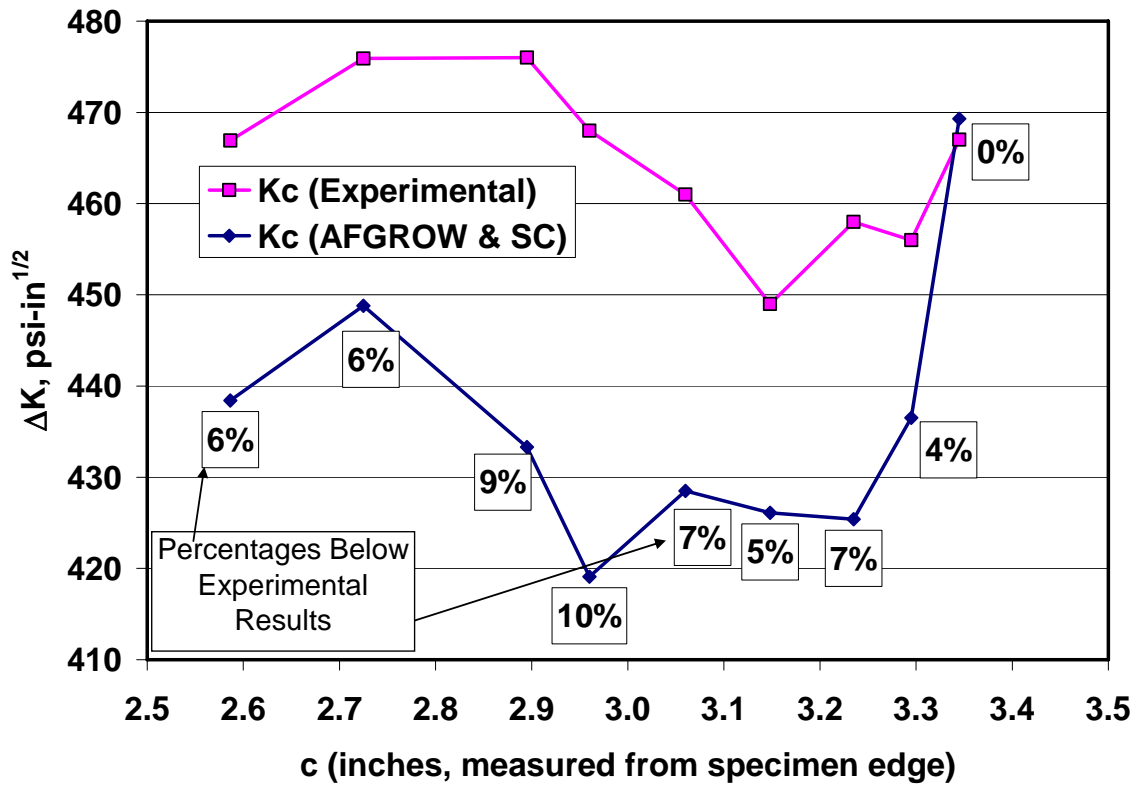
Figure 10. Comparison of Computed SIFs to Experimentally Estimated SIFs.

2.2.4 Integral Structure—Two-Bay Crack

This structure is similar to the Continuing Damage scenario in the Integral Structure described in the previous section—the Integral Structure models served as the starting point for the two-bay crack model, Figure 11. The crack can extend into either of the two bays shown in Figure 11, and the user can stop crack propagation in either of the two bays. Table 6 below Figure 11 describes the parameters and the valid ranges.
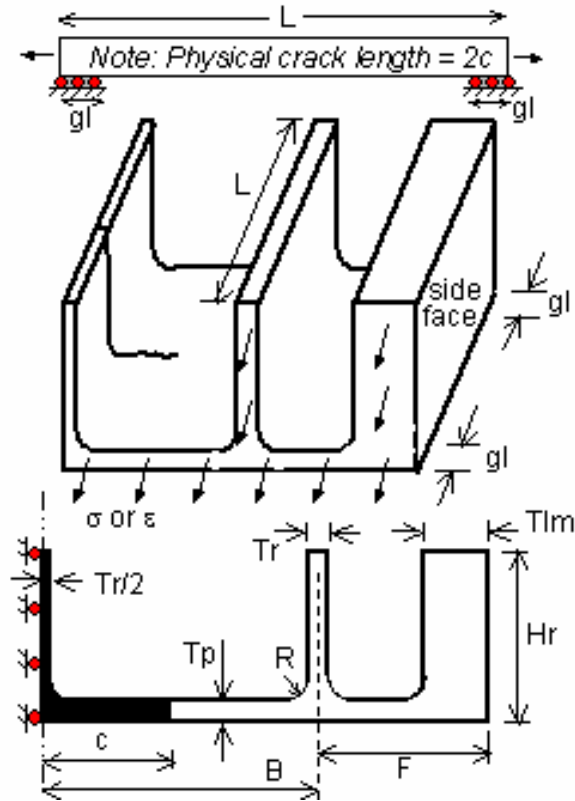
Figure 11. Integral Structure—Two Bay Crack (side view, trimetric view, and cross-section at crack. Plane of symmetry and central broken stiffener are shown in the lower view.

Table 6. Range of Parameters for Integral Structure—Two-Bay Crack.

Parameter	Description	Range
a	Crack Bore Length	**Tp**≤**a**≤**Hr**
B	Distance of Stiffener from Hole	3**Tp**+**D**/2+**R**≤**B**≤10
c	Half-Crack Length	**Tr**/2+**R**+0.03≤**c**≤ **B**+**F**-**Tlm**-**R**-0.03
F	Distance of Stiffener from Edge Away from Hole	**Tr**/2+2**R**+**Tm**+0.03≤**F**≤**Tr**/2+2**R**+**Tlm**+20
gl	Grip Length	0≤**gl**≤**L**/2
Hr	Stiffener Height	**Tp**+**R**+0.25≤**Hr**≤**Tp**+**R**+5
L	Panel Length	**B**≤**L**≤4(**ED**+**B**+**F**)
lc	Local Ellipse Axis (Stage 5)	0≤**lc**≤**c**
Modulus	Young Modulus	No limit
Nu	Poisson Ratio	0≤**Nu**<0.5
R	Fillet Radius	0.125≤**R**≤0.5
Tlm	Lumped Mass Thickness	**Tr**≤**Tlm**≤fcn(**F**)
Tp	Panel Thickness	0.1≤**Tp**≤0.5
Tr	Stiffener Thickness	0.1≤**Tr**≤0.5

The user can specify a constant stress on the boundary opposite the crack face. Similar to the continuing damage model, the application of a grip pressure on the top and bottom of the part is modeled with a fixed normal displacement over a specified grip length on the bottom of the plate. The actual length of the modeled integral structure can also be controlled by the user.

2.2.5 Challenges

Several modifications to AFGROW were needed to implement the Interactive Solution Strategy. Since many of these modifications were also needed to implement the other Strategies, for conciseness, all AFGROW modifications are summarized in Section 3.3 of this report.

In addition to the challenges associated with the AFGROW modifications discussed in Section 3.3, several challenges unique to the integral structure models were overcome to implement the Interactive Solution Strategy. A significant challenge was to estimate a realistic crack growth scenario—the integral structures were unique and therefore untested, so that the crack growth scenarios (where the cracks tended to propagate and the basic crack shapes) were unknown. Even if it is assumed that the cracks propagate in one plane, even through the upright stiffeners, one does not know whether the crack is straight, elliptical or some other shape as the cracks grow through the plate and stiffeners. Another significant challenge that was overcome is the on-the-fly three-dimensional modeling; as the crack propagates, the mesh must be automatically (that is, without user input) updated to maintain finite element mesh integrity and numerical accuracy. Many automatic meshing algorithms are not very robust, and numerical results can be suspect if finite elements get too distorted. ESRD, Inc., the developers of StressCheck®, implemented through a licensing agreement with another company a robust automeshing algorithm that allows the solution to proceed automatically, without intermediate user inputs. Even so, the automeshing has unavoidable "dead

zones", or combinations of parameters where an automesh cannot be created.

2.2.6 Limitations

As mentioned above, several combinations of parameters undoubtedly exist where the automesher will be unable to generate a mesh. In such cases, the K solution(s) will revert to the prior value and the crack front(s) propagated another interval. Obviously, if a user attempts to begin an analysis in a 'dead zone', there will be no prior K solution and the analysis will fail.

Implementation of the Interactive Solution Strategy can be tricky; the integrator has to be intimately familiar with the fundamental characteristics and operations of both the External K-solver, as well as with the AFGROW crack growth analysis software and Microsoft COM programming, or be able to work closely with somebody who can fill in expertise gaps. Therefore we recommend that only experienced External K-solver and AFGROW personnel should perform the modeling and meshing of the multitudes of stages that will be required and dictate the switching of models during the assessment.

2.2.7 Recommendations

The feasibility and utility of the Interactive Solution approach has been demonstrated. While the list of geometry, loads, and crack growth scenario combinations is theoretically infinitely long, practically, limitations on computing horsepower needed for any arbitrarily complicated, full three-dimensional problems will always shorten the list considerably. However, the list of practical, immediately attainable solutions to real aircraft problems still is very long, a partial list is shown below:

- Multiple Site Damage at Columns of Holes
- Multiple Site Damage at Patterns
- Multiple Site Damage at Lap Joints
- Multiple Site Damage at Step Joints
- Web to Stiffener Cracking
- Web to Flange Cracking in Integral Structure
- Reinforced Lug to Web
- Skin Panels with Stiffeners
- 3-D Castings
- Web to Stiffener Cracking
- Shear Joints and Stepped Shear Joints
- Stringer Run-outs
- Skin to Spar Cracking
- Two-Bay Scenarios
- Durability Criteria for New Designs, e.g. JSF, etc.
- Engine Components — Vibratory Cracking

In addition to the technical aspects of the implementation of the integrated K-solver—AFGROW software, there are regulatory issues that must be addressed, as they can be just as important as the technical issues. For instance, though the integral or unitized structure has tremendous promise for reducing weight and cost of aircraft structure, to take full advantage of this considerable savings, new damage tolerance acceptance criteria must be developed in coordination with the OEM and the regulatory agencies to get full 'buy-in' from all relevant parties to fully realize the benefits.

2.3 AFGROW External K-Solver: Handbook Solutions

2.3.1 Objective

The Handbook Solution strategy will be demonstrated by fully integrating the External K-solver with AFGROW software for two-dimensional notches of various type and dimensions. AFGROW will call the External K-solver only when it needs Stress Intensity Factors (SIFs) for its crack growth calculations. Crack dimensions will be updated in the External K-solver and updated SIFs will be passed to AFGROW as the crack growth calculations continue. The AFGROW-External K-solver interaction continues until AFGROW (through Preferences input by the user) decides to stop the crack growth; crack growth can stop due to various criteria selected by the user. The Handbook Solution strategy differs from the Interactive Solution strategy in its complete reliance on understanding the desired parameter ranges and construction of a robust set of External K-solver models prior to integration with AFGROW; no automeshing is required as in the previous Solution Strategy.

2.3.2 Description

The term "Handbook" is derived from the StressCheck® External K-solver nomenclature. Handbooks, as StressCheck® uses them, are specific structural geometry, load and crack configurations that are fully parameterized to allow the user to extract many engineering data or solutions for the similar configurations quickly, much as classical engineering handbooks such as Roark's Formulas for Stress and Strain, Reference [4] are used to extract engineering data for particular structural geometry and loads combinations. StressCheck® has an interface that allows expert finite element method users to construct models so that less-than-expert users can run trade studies and sensitivity studies on a large number of variations of parameters for the constructed models. For instance, an expert user could set up a notched structural geometry in the Handbook interface so that a designer could examine the effect of variations in many parameters on the stress concentration in a notch for a particular loading scenario. In the context of this project, Handbook then refers to relatively simple geometry, load and crack configurations that are nonetheless quite useful and powerful to the designer. Several such useful and powerful Handbooks were constructed and made available on the external-K server.

There are many problem types, several candidate examples of which are described and implemented in this report, that are key to damage tolerance assessments on aircraft structures. While AFGROW contains many useful solutions, the intrinsic library was greatly expanded by using Handbook methods described in this report. Models were built and parameters set up by appropriately trained External K-solver analysts, such as Boeing, AFRL, and APES personnel. Several Handbook cases have been integrated into AFGROW with additional menu options within a separate geometry input screen, and the solutions can be obtained interactively with the External K-solver. Acceptance criteria, checks on model parameter ranges, and solution checks were made by experienced finite element analysts and fracture experts. Error checks were made to validate Stress Intensity Factors for the entire specified range of parameters.

Robust External K-solver models for four (4) notch types were constructed, checked, validated, and integrated with an AFGROW 'beta' software by APES, and reside on the external-K server: Through Crack at a Semicircular Notch, Through Crack at a U-shaped Notch, Through Crack at a Slotted Notch, and Through Crack at a V-Shaped Notch. These four notch types will be described below in some detail. In addition, the Boeing Corporation constructed, check, and validated models for several combinations/locations of through crack(s) at Plate Nut holes (AKA "Nut Plate"). APES was responsible for integrating the Nut Plate models with AFGROW on the external-K server.

Similar to the Interactive Solution Strategy, the Handbook Solution Strategy used AFGROW integrated with the External K-solver—this integration can be accessed through the AFGROW Graphical User Interface (GUI) residing on an Internet accessible external-K Server by using the Remote Desktop Connection through Windows Terminal Services. Geometry and loads are controlled by the user through the AFGROW GUI. AFGROW communicates with the External K-solver via DLL's using Microsoft COM technology. The program integration includes all functions and methods necessary for setting and retrieving geometric, cracking, loading, and constraint properties, as well as returning Stress Intensity Factor(s), numerical convergence information, and error messages. The COM server is also responsible for selecting the appropriate StressCheck$^®$ models from the mesh library that provides the best convergence for the current cracked state and geometric parameters. APES contracted with LexTech, Inc. to perform the final stages of integrating the COM application(s) into the AFGROW code. The current AFGROW framework is used for setting crack propagation limits, defining output requirements, and visualizing the crack progression.

2.3.3   Through Crack at a Semicircular Notch

This geometry is also known as "Single Edge Notch-Tension" or SENT for short. AFGROW currently contains a model for the Through Crack at a Semicircular Notch. However, the radius of the notch is restricted to $1/16^{th}$ of the width of the section. The Through Crack at a Semicircular Notch models implemented on the AFGROW server eliminate this dimension restriction and expand this range of valid dimensions considerably. Theoretically, the dimensions can be expanded to almost any value; practically, the valid parameter range has been restricted, albeit to a much wider range than is currently available in AFGROW. Parameter definitions are shown in Figure 12 below. Table 7 below Figure 12 describes the parameters and the valid ranges. If the analyst chooses to use parameter values outside range indicated in the table, numerical accuracy in the finite element models might suffer unacceptably.
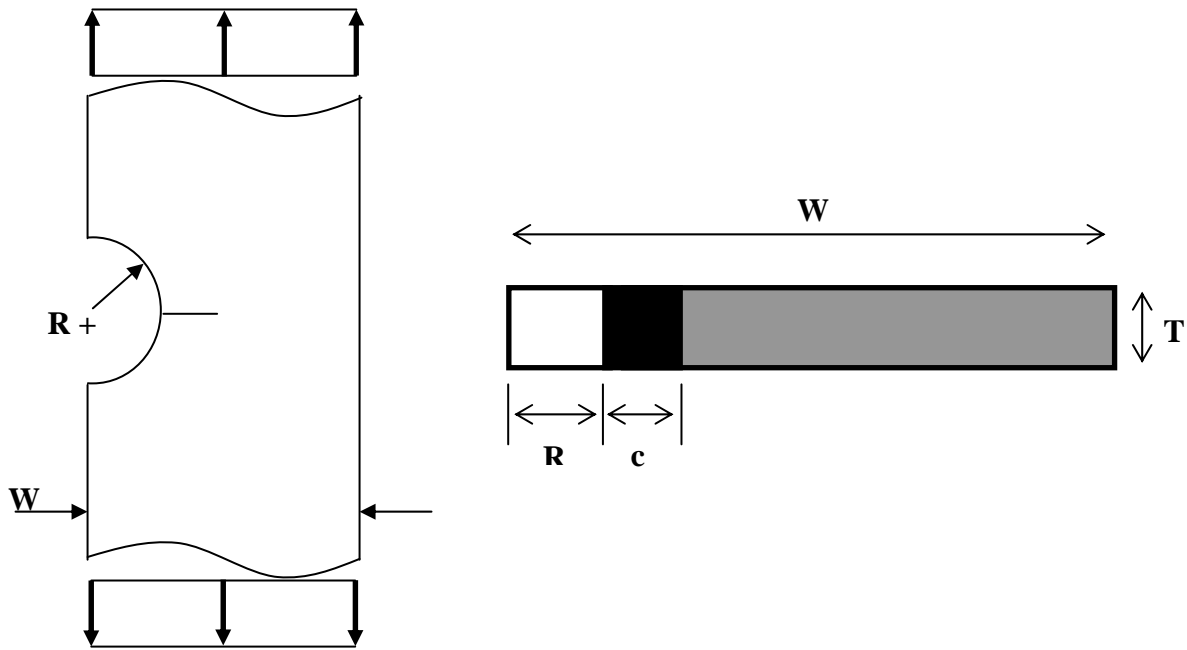
Figure 12. Through Crack at a Semicircular Notch. Uniform Tension is Applied at Far Field.

Table 7. Range of Parameters for Through Crack at a Semicircular Notch.

Parameter	Description	Range
W	Plate Width	$1 \leq \mathbf{W} \leq 10$
R	Notch Radius	$0.0625\mathbf{W} \leq \mathbf{R} \leq 0.8\mathbf{W}$
c	Through Crack Length	$0.005\mathbf{W} \leq c \leq 0.8(\mathbf{W}\text{-}\mathbf{R})$
T	Plate Thickness	No limit

2.3.4 Through Crack at a U-shaped Notch

Parameter definitions for the Through Crack at a U-shaped Notch are shown in Figure 13 below. Table 8 below the Figure describes the parameters and the valid ranges. If the analyst chooses to use parameter values outside range indicated in the table, numerical accuracy in the finite element models might suffer unacceptably.
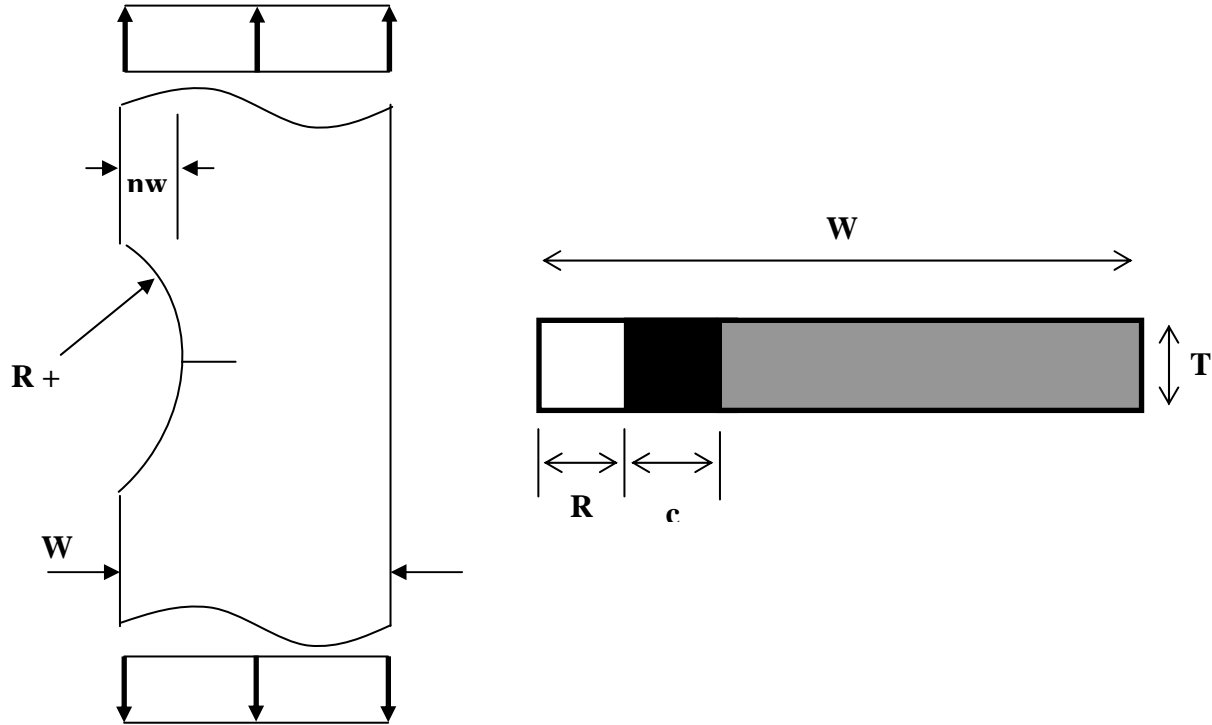
Figure 13. Through Crack at a U-shaped Notch. Uniform Tension is Applied at Far Field.

Table 8. Range of Parameters for Through Crack at a U-shaped Notch.

Parameter	Description	Range
W	Plate Width	$1 \leq W \leq 10$
R	Notch Radius	$1.105 nw \leq R \leq (4W + nw^2)/(2\ nw)$
c	Crack Length	$0.005 W \leq c \leq 0.8(W - nw)$
nw	Notch Width	$0.1 W \leq nw \leq 0.8 W$
T	Thickness	No Limit

2.3.5 Through Crack at a Slotted Notch

Parameter definitions for the Through Crack at a Slotted Notch are shown in Figure 14 below. Table 9 below the Figure describes the parameters and the valid ranges. If the analyst chooses to use parameter values outside range indicated in the table, numerical accuracy in the finite element models might suffer unacceptably.
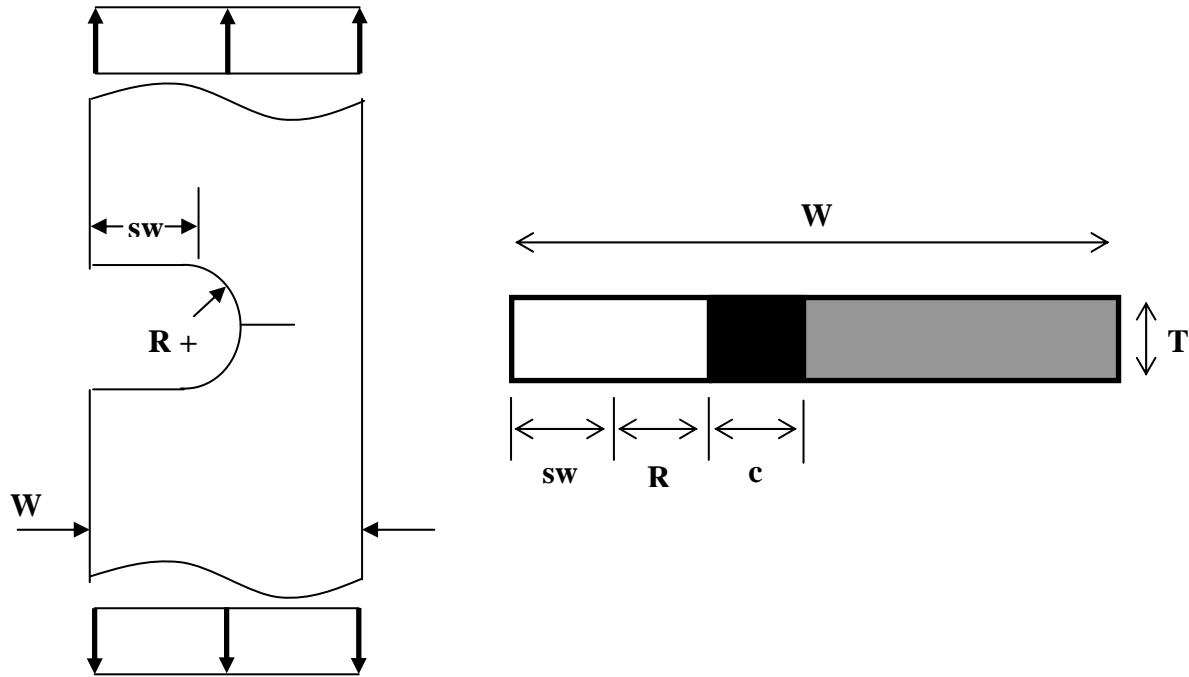
Figure 14. Through Crack at a Slotted Notch. Uniform Tension is Applied at Far Field.

Table 9. Range of Parameters for Through Crack at a Slotted Notch.

Parameter	Description	Range
W	Width	1≤**W**≤10
R	Notch Radius	0.0625**W**≤**R**≤0.77**W**
c	Crack Length	0.005**W**≤**c**≤0.80*(**W**-**R**-**sw**)
sw	Slot Width	0.03**W**≤**sw**≤0.80***W**-**R**
T	Thickness	No Limit

2.3.6 Through Crack at a V-Shaped Notch

Parameter definitions for the Through Crack at a V-shaped Notch are shown in Figure 15 below. Table 10 below the Figure describes the parameters and the valid ranges. If the analyst chooses to use parameter values outside range indicated in the table, numerical accuracy in the finite element models might suffer unacceptably.
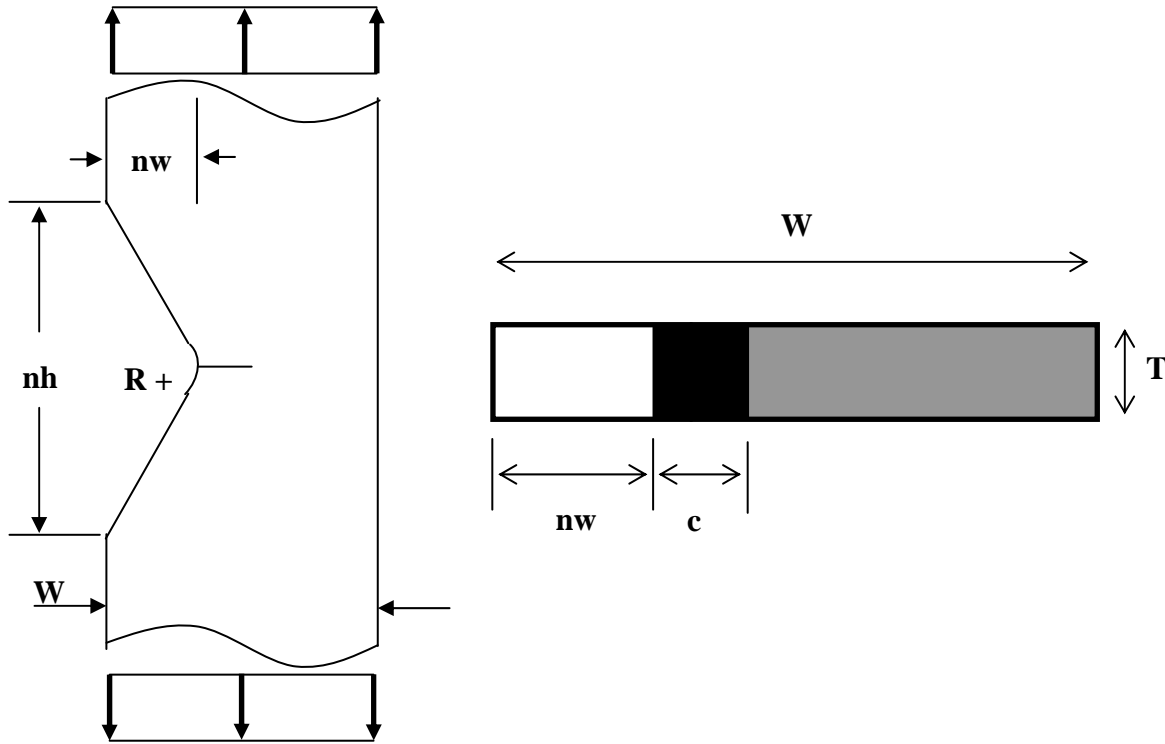
Figure 15. Through Crack at a V-Shaped Notch. Uniform Tension is Applied at Far Field.

Table 10. Range of Parameters for Through Crack at a V-Shaped Notch.

Parameter	Description	Range
W	Width	1≤**W**≤1
R	Notch Radius	0.10≤**R**≤1.0
c	Crack Length	0.005≤**c**≤0.80(**W-nw**)
nh	Notch Height	1.1**R** ≤**nh**≤2
nw	Notch Width	0.03≤**nw**≤0.8
T	Thickness	No Limit

2.3.7 Through Crack at a Plate Nut

The Boeing Corporation was responsible for construction and of all models in this category. However, APES is still responsible for accuracy, limits, results, and practicality checks, as well as implementation with AFGROW. Schematics of the loading conditions for all of the plate-nut models are found below in Figure 16; fourteen (14) different geometry and balanced load configurations have been implemented within the AFGROW beta version. Each of these models can be superimposed with the others, allowing the user flexibility in defining the load and reaction condition. The user can pick the desired cracking scenarios from the choices shown in Figure 17. The nut plate is constrained in the X and Y directions with a two-point rigid body constraint.
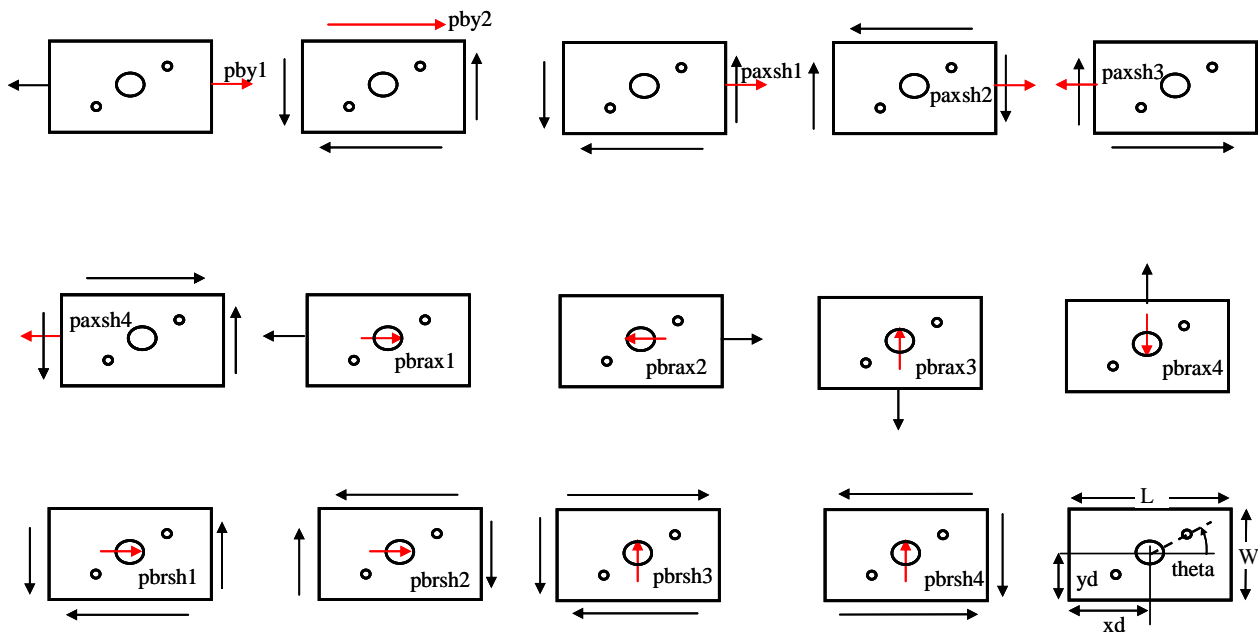
Figure 16. Rotated Nut Plate Geometry, Loads and Constraints. All Models Two-Dimensional.

| 1. Single crack originating from a satellite hole | 2. Single crack originating from the main hole | 3. Ligament broken between the main hole and a satellite hole, and a single crack originating from the satellite hole. | 4.Two cracks originating from the main hole | 5. Ligaments broken between the main hole and the satellite holes, and a single crack originating from each satellite hole. | 6. Single crack originating from a satellite hole, growing towards the main hole |

Figure 17. Rotated Nut Plate Through Crack Scenarios.

The parameters as denoted in Figures 16 and 17 above can be varied within the limits shown in Table 11 below. In contrast with other models in this report, there are very few restrictions on the parameter space.

Table 11. Range of Parameters for Rotated Nut Plate

Parameter	Description	Range
L	Plate Length	No Limit
W	Plate Width	No Limit
D	Main Hole Diameter	Geometric
xd	Main Hole X-Center	Geometric
yd	Main Hole Y-Center	Geometric
ld	Satellite Hole Diameter	Geometric
theta	Satellite Hole Angle (degrees)	Geometric
rh	Satellite-to-Main Hole Center Distance	Geometric
th	Plate Thickness	Geometric
c	Initial Crack Length	$0.005 \leq c \leq$ Geometric
Modulus	Young's Modulus (ksi)	No Limit
pby1	Bypass Force, x-direction (lb)	No Limit
pby2	Shear Bypass Force, Top Face (lb)	No Limit
paxsh1	Force on positive x-face, reacted in shear on bottom face (lb)	No Limit
paxsh2	Force on positive x-face, reacted in shear on top face (lb)	No Limit
paxsh3	Force on negative x-face, reacted in shear on bottom face (lb)	No Limit
paxsh4	Force on negative x face, reacted in shear on top face (lb)	No Limit
pbrax1	Bearing Force in positive x-direction, reacted axially on left face (lb)	No Limit
pbrax2	Bearing Force in negative x-direction, reacted axially on right face (lb)	No Limit
pbrax3	Bearing Force in positive y-direction, reacted axially on bottom face (lb)	No Limit
pbrax4	Bearing Force in negative y-direction, reacted axially on top face (lb)	No Limit
pbrsh1	Bearing Force, x-direction, reacted in shear on bottom face (lb)	No Limit

Parameter	Description	Range
pbrsh2	Bearing Force, x direction, reacted in shear on top face (lb)	No Limit
pbrsh3	Bearing Force, y-direction, reacted in shear on left face (lb)	No Limit
pbrsh4	Bearing Force, y direction, reacted in shear on right face (lb)	No Limit

The External K-solver, StressCheck®, uses its "Crack Path" capability to create the finite element meshes as they are needed during the AFGROW crack growth analysis, but maintains the crack growth along a single plane (i.e., no crack 'turning' is allowed) determined by the uncracked principal stress state. Due to issues with multiple cracks and the crack path functionality, only the first three scenarios of Figure 17 have been implemented on the external-K server at this time. The remainder of the scenarios will be implemented as soon as the external-K solver software on the server can be upgraded. Additional issues unique to the plate-nut Handbook solutions are discussed in the Challenges and Limitations sections.

2.3.8   Multi-Site Damage (MSD)

This Handbook model demonstrates the flexibility of the AFGROW and External-K solver in handling multiple crack problems. Figure 18 shows a schematic of the MSD Handbook problem class integrated with AFGROW, the available parameters are the six crack lengths c1, c2, c3, c4, c5, and c6. This demonstration used an existing StressCheck handbook model (a model distributed with the StressCheck installation) as the basis for the AFGROW-External-K solver interaction. This model does not have any flexibility in the geometry of the uncracked state, so Figure 18 shows the fixed values used for the uncracked geometry, defined by W, L, S, ED, and D. The only parameter limits are the maximum allowable values for the input crack lengths, no crack length may be larger than the ligament width minus a small tolerance (S - D - 0.004" = 0.39"). Very small crack sizes may be input, but solution convergence will suffer. The solution quality reported in the AFGROW status bar is an average of the solution quality for all six crack fronts. The crack propagation stops when any crack hits its limiting length of 0.39 inch.

2.3.9 Challenges
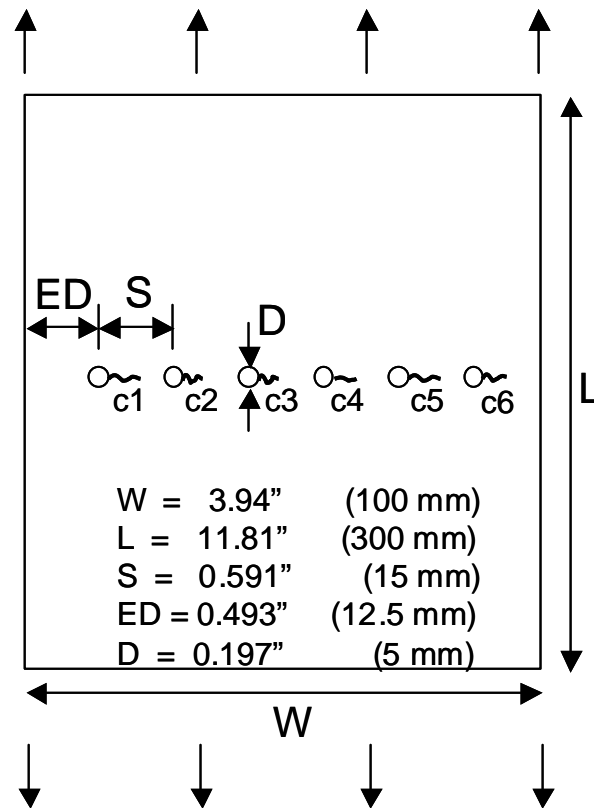
Figure 18. Multi-Site Damage (MSD) fixed geometry and crack length parameters.

At first glance, Handbook problem classes appeared to be a very easy strategy to implement; in fact, the integration is similar to the implementation in the Interactive Solution strategy; the significant difference being when was the finite element mesh defined: during the crack growth analysis (Interactive) or *apriori* (Handbook). In practice, we had to be very careful to define the desired parameter space (range of parameters that could be analyzed) as soon as possible when developing these Handbook models. Since we were always very concerned with numerical accuracy, we had to create several Handbook models for each of the models described in this report—some Handbook models were needed to handle 'small' cracks, other Handbook models were needed for intermediate sized cracks, and still other Handbook models were needed to handle the 'large' cracks or cracks that approached edges or holes. Quite a bit of testing was performed to ensure that numerical accuracy would not drop below 2% for any possible choice of parameters, so that the full range of parameters defined in their respective tables can be chosen by the user, and sufficient numerical accuracy obtained.

The implementation and interaction of each of the Handbook problem classes with AFGROW is consistent, except for the plate nut models. Several items are unique within the plate-nut class of Handbook problems. First of all, a plan-form drawing of the geometry is used, as opposed to a cross-sectional view of the cracking plane. This was done because the plane of cracking for all plate-nut models depends on the applied parametric load inputs. When the user starts an analysis of a plate-nut problem, the loads are combined, applied to the finite-element model in the external-K solver, and the principal stress angles are computed at either the main hole or the satellite hole. The crack(s) is

then forced to propagate perpendicular to these principal stress angles. Consequently, when a user inputs an initial crack length for these problems, it is not displayed on the geometry in the AFGROW window until the analysis is begun.

The plate-nut solutions are also unique within the Handbook framework because of issues existing with the Crack Path automesher in the current server version (6.2.2k) of StressCheck®, the issue being that longer initial crack lengths cannot be explicitly set for the plate nut models. SIF's for the longer crack lengths must be obtained by beginning with a small crack length (i.e., less than the diameter of the hole) and allowing the external-K solver to propagate the crack automatically and incrementally. This issue is what prevents the multiple-crack plate nut scenarios from being implemented, as it is impossible to explicitly set any given combination of two longer crack lengths in the external-K solver in order to obtain the SIF's. To work around this issue using StressCheck version 6.2.2k, the three single-crack plate-nut models are used as follows: after the principal stress angles are determined, the SIF's for a *table* of crack lengths are obtained. The initial crack length and the number of increments obtained are fixed in the software that performs the AFGROW and external-K solver integration. Then, using the initial crack length specified for the analysis by the user, the SIF's are obtained by interpolation of this table. In practice, compared to the other Handbook problems, the analysis will take a while to begin the crack propagation (as the table is built), but then the analysis will progress rapidly as each successive SIF need only be looked up in a table.

Overall, several modifications to AFGROW were needed to implement the Handbook Solution Strategy. Since many of these modifications were also needed to implement the other Strategies, for conciseness, all AFGROW modifications are summarized in Section 3.3 of this report.

2.3.10 Limitations

If numerical accuracy is a consideration, each model is limited by parameter range specified in the appropriate table. However, in keeping with the usual AFGROW operation, very few actual limits on the crack growth analysis are imposed; as long as numerical accuracy is ignored, theoretically the only limits are physical ones, for instance, a notch can't be deeper than the width of the entire domain, or a hole cannot be so close to an edge that the hole is cut off, resulting in a notch. For the plate nut scenarios, the current implementation of the three single-crack models is limited by the resolution of the SIF lookup table discussed in section 2.3.9. This limitation can be alleviated when the server is upgraded to a more recent version of StressCheck, as any combination of crack lengths can be set explicitly in recent versions of the external-K solver.

2.3.11 Recommendations

The feasibility and utility of the Handbook Solution Strategy has been demonstrated. While the list of geometry, loads, and crack growth scenario combinations is theoretically infinitely long, practically, limitations on computing horsepower needed for any arbitrarily complicated, full three-dimensional problems will always shorten the list considerably. However, the list of practical, immediately attainable solutions to real aircraft problems remains significant:

- Shielded Through Crack Around Fasteners
- Shielded Part-through Cracks
- Shielded through Cracks around Cut-outs and Doors
- Shielded through Cracks around Reinforced Geometry
- Continuing Damage at Fastener Holes
- Cracks growing to Fastener Holes, including Processed Holes
- Multiple Site Damage at Columns of Holes
- Multiple Site Damage at Patterns
- Multiple Site Damage at Lap Joints
- Multiple Site Damage at Step Joints
- Lug Base Transition
- General Geometry: Stringers
- General Geometry: Skin Panels
- Landing Gear
- Exfoliation of Skin on Wing Surfaces
- Web to Stiffener Cracking
- Skin to Spar Cracking
- Two-Bay Scenarios
- Joints and Splices
- Durability Criteria for New Designs, e.g. JSF, etc.
- Vibratory Cracking Engine Component
- Spectrum Benchmarking
- Compression Dominated or High Fastener Load
- Interference Fasteners or Cold Working

To keep the number of handbook models 'small' and workable, the analyst still must be very careful up front to define as accurately as possible the desired parameter space (that is, the range of parameters needed to define the desired geometry).

2.4 Table Lookup Solutions: Part-Through Crack in Lug

2.4.1 Objective

The Table Lookup Solution strategy is demonstrated by obtaining detailed tables of Stress Intensity Factors or Geometric Factors β for three-dimensional lugs with part-through corner cracks from the External K-solver, then fully integrating these tables with the AFGROW software. AFGROW will call the tables whenever it needs Stress Intensity Factors (SIFs) for its crack growth calculations. Spline interpolation routines created by the AFGROW developers will be used to interpolate between the 'hard points' in the tables. Crack dimensions will be updated in the External K-solver and updated SIFs will be passed to AFGROW as the crack growth calculations continue. The AFGROW-External K-solver interaction continues until AFGROW (through Preferences input by the user) decides to stop the crack growth; crack growth can stop due to Net Section Yield, specified Maximum Crack Length or K_{max} reached, among other criteria.

2.4.2 Description

Lugs can be found in many different substructures in an aircraft: wing attach fittings and control linkages among others. AFGROW currently contains a model for the Part-through Crack in Lug. However, users of this AFGROW crack growth model in the past have indicated that the results from the AFGROW analyses can be suspect; therefore, obtaining accurate and reliable Stress Intensity Factors for part-through cracks in lugs was a major goal of this project.

Hundreds of three-dimensional finite element models were constructed and solved to model part-through cracks in lugs for a wide range of parameter variations, Figure 19 and Table 12. During the finite element solution, Stress Intensity Factors (SIFs) were extracted at several points along each part-through crack front. AFGROW uses only two SIFs to drive part-through cracks; therefore only the two 'local maxima' at the two crack tips were assembled into tables. These 'local maxima' were selected by ignoring the two points closest to the boundaries (0 degrees and 90 degrees along the crack front) and then searching for the maximum within 10 degrees of the appropriate boundary. The SIFs tables were accurate to within 2%. SIFs were non-dimensionalized to create Geometric Factors, β. The space of all pre-specified, allowable parameter ranges was spanned by these tables of Geometric Factors. These tables of Stress Intensity Factors or Geometric Factors β from the External K-solver are being integrated with the AFGROW software by the AFGROW developers. AFGROW querys the tables whenever it needs Stress Intensity Factors (SIFs) for its crack growth calculations; as the tables contain Geometric Factors at distinct values of the parameters, intermediate values of parameters are used to interpolate between the tables when needed.

The tabular lookup solutions will be fully integrated and distributed with the AFGROW software, no external-K solver is required (in real-time) to run a crack growth analysis using these solutions. This strategy is ideal for portability and distribution of the SIF's with AFGROW, but is limited by the fact that multi-dimensional interpolation must be used to obtain SIF's between 'hard points' of the table, which can lead to errors if the quantity of 'hard points' is not sufficient.
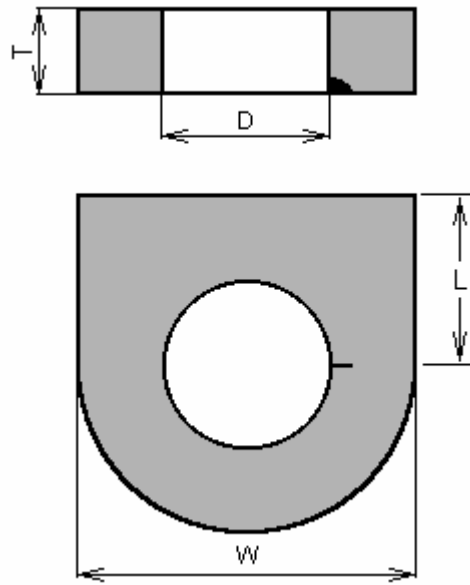
Figure 19. Part-through Crack in Lug.

Table 12. Range of Parameters for Part-through Crack in Lug.

Parameter	Description	Range
a	Bore Crack Length	$0.005T \leq a \leq 0.9T$
c	Surface Crack Length	$0.005 \leq c \leq 1.8$
D	Hole Diameter	$0.25T \leq D \leq 4T$
T	Thickness	No limit
W	Lug Outer Diameter	$1.3D \leq W \leq 5D$

Acceptance criteria and checks on models and solutions were made by experienced finite element analysts and fracture experts. Error checks were made to validate Stress Intensity Factors for the entire specified range of parameters. A summary of solution quality for this family of solutions is described in the following section.

2.4.3 Challenges

Over 600 three-dimensional (3D) finite element (FE) models were needed to construct part-through cracks in lugs for a wide range of parameter variations—many of these models were created during the natural trial-and-error process that results when refining mesh strategies to meet stringent accuracy goals; in the final tally of all models needed to span the entire range, only 8 models were needed. Nevertheless, this few models represent several months of development by an experienced finite element analyst. The solution quality estimate (the percentage difference between the best external-K solver solution and a solution extrapolated to an infinite number of degrees of freedom using a Richardson extrapolation of the three best p-level solutions) in general, was excellent. Of the 635 total solutions, 586 have a solution quality estimate less than 1%, 42 have a solution quality estimate between 1% and 2%, and 7 have a solution quality estimate between 2% and 2.5%. The 49

solutions above 1% are generally at geometric parameter combinations that would be considered extreme, or atypical of standard industry design practice.

Since fracture mechanics parameters are calculated from a finite number of tables, interpolation and extrapolation are very important considerations—clearly you want the interpolation and extrapolation error to be 'small', however large you define 'small' error to be. There were actually two notions to consider when choosing the interpolation scheme: do you choose linear or higher order interpolation; and which 'hard points' will be used to define the curves which are used in the interpolation. 'Hard points' are the specific parameter variations where finite element solutions were run and whose results were collected in the tables that are to be used in the interpolations. For instance, the 'hard points' in the tables are for the ratio of the lug width to the hole diameter are $W/D = 1.5$ and $W/D = 2.5$ —all data in the tables is defined at either $W/D = 1.5$ or $W/D = 2.5$. In a more extreme case, for the ratio of the bore crack length to the lug thickness, A/T, 'hard points' were defined at 0.01, 0.05, 0.1, 0.2, 0.4, 0.6, 0.7, 0.8, 0.9—a total of 9 A/T hard points were necessary to keep the interpolation error below 5%.

Several modifications to AFGROW were needed to implement the Interactive Solution Strategy. Since many of these modifications were also needed to implement the other Strategies, for conciseness, all AFGROW modifications are summarized in Section 3.3 of this report.

2.4.4 Limitations

The user can vary the parameters in the Part-through Crack in Lug models over theoretically infinite range—if fracture mechanics data is needed for geometries outside the range of the geometry combinations that were used to define the tables in the Table Look-up, AFGROW merely extrapolates using the defined tables. However, if the analyst chooses to use parameter values outside range indicated in the table, numerical accuracy in the finite element models might suffer unacceptably.

Initially this lug geometry was thought to be relatively simple and it was estimated that it would take only about a month to create enough tables to adequately populate the matrix of data. However, after much testing, and verifying numerical accuracy of the results, as well as taking into account feedback from other project partners, it was clear that many more solutions would be necessary that originally projected. In the initial capability, a total 635 3D finite element solutions of the lug with part-through crack were necessary to obtain reasonably accurate interpolations. This total takes into account the range of variables requested by project partners, as well as desired accuracy in the quadruply interpolated tables.

2.4.5 Recommendations

The feasibility and utility of the Table Look-up Solution Strategy has been demonstrated. While the list of geometry, loads, and crack growth scenario combinations is theoretically infinitely long, practical limitations on computing horsepower needed for any arbitrarily complicated, full three-dimensional problems will always shorten the list considerably. However, the list of practical, immediately attainable solutions to real aircraft problems still significant:

- 3-D Fittings—3D fittings are in many places in an aircraft and in machinery; wing attach points, floor fittings, door hinges, etc. Due to their complexity, special care will be needed, especially at the beginning of creating the tables needed for the Look-up procedure, to minimize the number of variables and hence keep the amount of time needed to do all the required finite element analysis time at a reasonable level.

- Lug Base Transition—Lugs are also in many places in an aircraft and in machinery; wing attach points, floor fittings, door hinges, etc. The initial lug analysis capability described here needed a minimum of 635 3D finite element solutions to adequately 'populate the matrix' of Stress Intensity Factors used in the Table Look-up interpolation procedures—this was for just one geometry of the base transition, in which the base was the same width as the section at the lug hole center. To define a model that allows a different base transition would require 635 3D FE solutions per each base— even if you allowed just one type of base—for instance, a symmetric transition to a wide base, you would need FE solutions at several transition angles to adequately populate the Table Look-up database. Again, due to their complexity, special care will be needed, especially at the beginning of creating the tables needed for the Look-up procedure, to minimize the number of variables and hence keep the amount of time needed to do all the required finite element analysis time at a reasonable level.

2.5 Additional Plug-In Modules for AFGROW

2.5.1 Objective

The Plug-In Module Solution Strategy, which essentially tests and proves the ability of AFGROW to adapt to evolving customer needs, will be demonstrated by integrating a new user-defined material crack growth rate option with AFGROW. Additionally, each specific implementation of the Handbook and the Interactive solution strategy within AFGROW has also become known as a 'Plug-In'. These implementations are further discussed in Section 3.3.

2.5.2 Description

AFGROW currently offers the user 5 different methods to incorporate material crack growth rate curves: Forman Equation, NASGRO Equation, Walker Equation, Harter-T method, and Tabular Look-up. The first 3 methods are well-known within the fracture mechanics community, so will not be described here. The Harter T-method allows the user to select from a very large menu of predefined crack growth rate curves for many different materials including aluminum and steel. The method uses a power law relationship to describe the stress ratio R variation in the crack growth rate, da/dN by using only one base Stress Intensity Factor Range, ΔK_I and one power law coefficient, m. The Tabular Look-up method, which also uses the Harter T-method developed by Jim Harter at AFRL/VA, allows a variable power law relationship to interpolate the crack growth rate, da/dN, across several stress ratios, R. The user is required to input crack growth rate versus the Stress Intensity Factor Range, ΔK_I at constant da/dN values. Often this is a very labor intensive task, to convert experimental data which might be sorted by R, where each R has a distinct set of da/dN - ΔK_I pairs. To improve on the flexibility of AFGROW in this area, a "da/dN Material Plug-in" will be developed which allows the user to define material behavior in any fashion possible via a programmatic interface.

The da/dN Material Plug-in capability will be developed by project partner Lextech, Inc.

2.5.3 da/dN Material Plug-in Tools

This new material "plugin" capability is similar to the User-defined "plugin" stress intensity and geometry models developed for the AFGROW framework. The capability allows the user complete freedom in defining the material crack growth rate curve; essentially this is to be an additional Material Property description in AFGROW, "User Defined", that calls COM components responsible for returning a crack growth rate, da/dN given a ΔK and an R-ratio. All interpolation, extrapolation, R-shifting, threshold, etc. are the responsibility of the COM component. The COM

component would also have to define yield, E, K_c, K_{Ic}, etc.

This approach allows- developers external to the AFGROW development team to create "material translators" from any source of da/dN data—the COM component allows the user to put the data into AFGROW tabular lookup form (a 'helper' algorithm), or the user could have AFGROW query the COM component directly for da/dN. The first way still allows the use of the AFGROW GUI to see and manipulate the data but can be complex for potential users to code; the second way can be implemented more rapidly by the user, but it may require the individual COM components to control their own GUI windows.

The following parameters are required to be defined by any material plugin COM interface that is developed as a material plugin for use by AFGROW, Table 13:

Table 13. Material Plug-in Parameters Required in COM Components.

Material Name

Coefficient of Thermal Expansion

Young's Modulus

Poisson's Ratio

Yield Stress

Ultimate Strength

Plain Strain Fracture Toughness

Plain Stress Fracture Toughness

Upper Limit On R shift

Lower Limit On R shift

Delta K Threshold at R=0

3   Deployment of External-K Solutions

3.1   Introduction

The system specification was described in detail in the document "Crack Growth and Stress Intensity Prediction Techniques, Volume 2: Feasibility Study for Interacting AFGROW With External K-Solvers," Final Report for Contract F33615-98-D-3210, Reference [1]. This report detailed general requirements for both the External K-solver and for the crack growth analysis software in order to implement an "integrated External K-solver—crack growth software" analysis tool using Microsoft Component Object Module (COM) technology. This section describes the specific capabilities developed to implement the system specification.

3.2   Windows Terminal Server

A Windows 2000 server dedicated to this project has been purchased and installed at APES. The machine is a dual-processor 2.8 GHz Athlon with 2GB of RAM in order to obtain interactive K-solutions as rapidly as current technology allows. StressCheck® has also been licensed and installed on this machine; external users may not make use of the StressCheck® GUI but may obtain K-solutions via StressCheck's COM server for specific, preconfigured problem types.

User access to the server is currently obtained via Windows Terminal Services (WTS). WTS allows a client computer to connect to the server in a terminal emulation mode, where the only things passed to the server are keyboard and mouse inputs and the only thing returned to the client is the display. Either the standard windows terminal client (under Windows 2000 or XP's Start Menu...Programs...Accessories...Communications...Remote Desktop Connection) or the Terminal Services Advanced Client (TSAC), a Microsoft plugin for Internet Explorer, can be used. Remote user logins, for interactive K-solver demonstration purposes, are restricted to run a single executable code (AFGROW) at startup. This executable code, via COM technology, allows full access to the beta AFGROW GUI using plugin technology and also to the limited, problem-specific plugin components (ActiveX DLL files) that obtain Stress Intensity Factors (SIFs) from StressCheck® models.

Client machines running either the Windows 2000 Professional operating system or the Windows XP Professional operating system have built-in Terminal Services Client Access Licenses (TS CAL's). Other operating systems will require a TS CAL, which can be installed by APES, and costs about $80 per client machine that needs access to the terminal server. Clients other than Win2000 Professional or WinXP Professional may access the Terminal Server on a trial basis. 90 days after the first access date, the trial license will expire and the client will no longer be able to connect to the terminal server without a TS CAL. Note that the TS CAL, once purchased, is very difficult to transfer between client machines. If a TS CAL is purchased for your client machine, please ensure that machine will be your primary method of access to the terminal server.

Information on how to access the external-K server via WTS can be found at http://externalk.dyndns.org/. Alternately, if a client machine has access to its own version of StressCheck® software, the AFGROW beta version and the DLL files controlling the interface with AFGROW and the external-K solver can easily be installed locally, avoiding the need to use WTS.

3.3   Implementation into AFGROW

The integrated External K-solver—AFGROW software is implemented into a beta version of AFGROW which resides on the server described in Section 3.2. During the execution of this project, a small terminology change was made—all integrated models are accessed through the Input/Model/Plug-In Model window; the term "Plug-In" now applies to all the integrated models as well as to the Material Plug-In model. The beta GUI looks much like the "production release" GUI, with the exception of the additional option under the Input/Model menu, Figure 20. Selection of this option launches another window that reveals all Plug-in Models the user can select for crack growth analysis, Figure 21. The models in the list of Figure 21 are controlled at AFGROW startup by a text-only configuration file, so that any user can customize which plugins are available in the list.
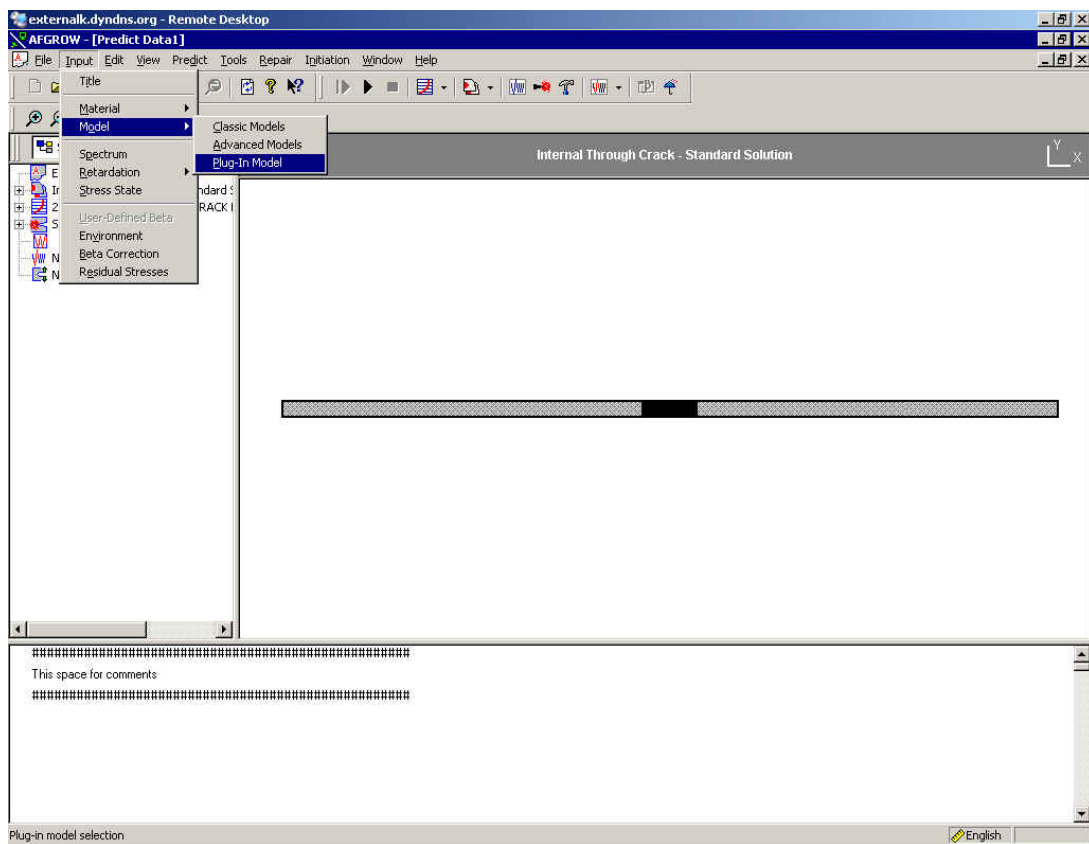
Figure 20. Select "Plug-in Model" in AFGROW beta Version to Start Modeling Process.

Once the model is chosen, the model is loaded along with a menu window that allows the user to adjust the geometry, load, constraint and other parameters from the default values, Figure 22. Because the External K-solver is software based on the *p*-version of the finite element method, the user can also choose polynomial or *p*-level for the analysis, $1 \leq p \leq 8$; the higher the p-level, the better the numerical results. The higher *p*-level comes at a price, of course; the higher the *p*-level, the longer the computation times. After selection of the model, the model schematic disappears, but can be referenced later by selecting the View/Preview Plug-in option.

44
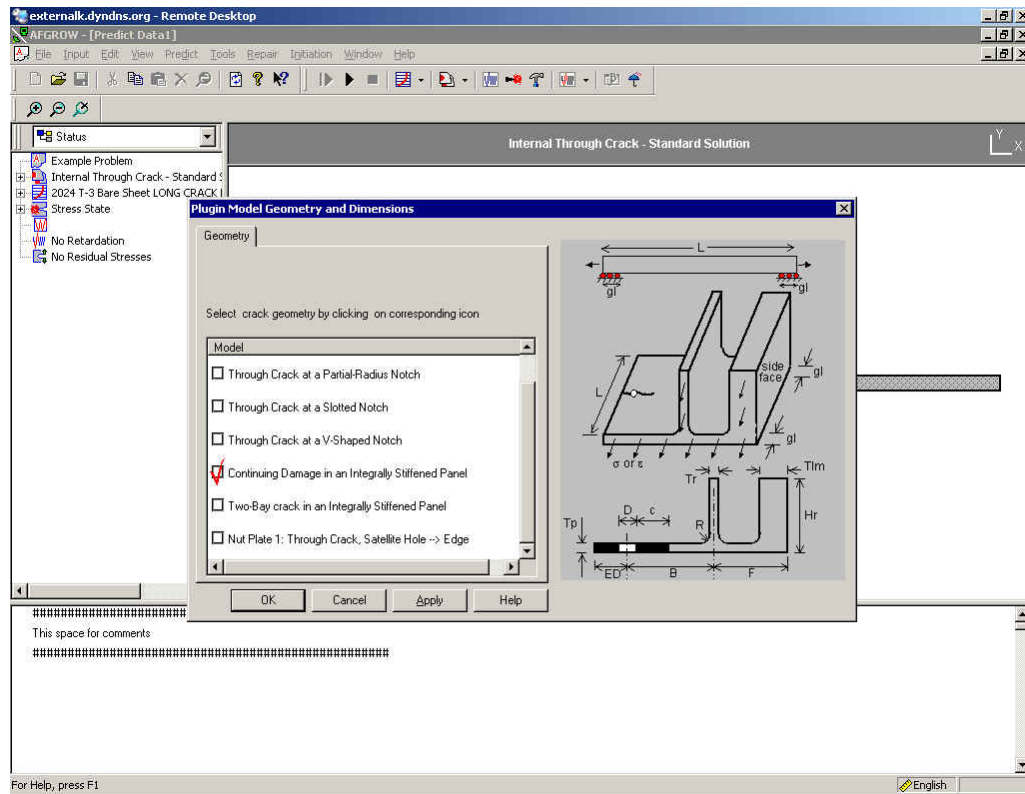
Figure 21. Additional Models Significantly Expand Number of Crack Models User Can Access.
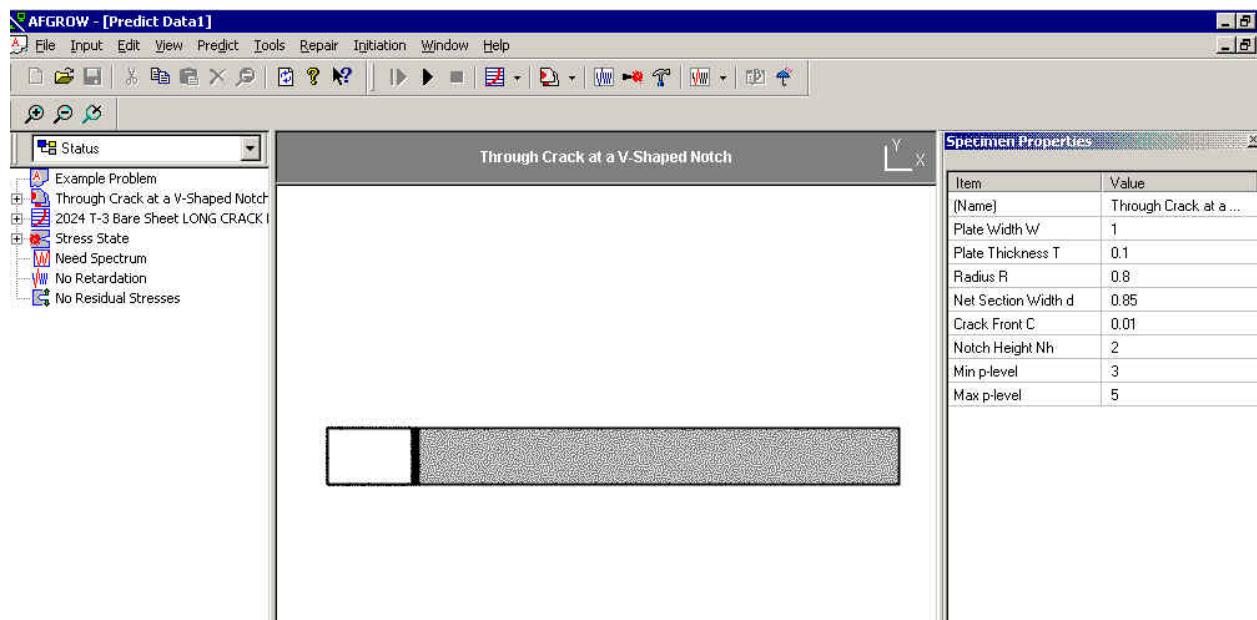
Figure 22. User Adjusts Geometry, Loads, Constraints and Polynomial Level in Specimen Properties Window (shown at right in this Figure) within the AFGROW GUI.

3.4 Summary of AFGROW Modifications

AFGROW modifications have been addressed as needed during the discussions of the Solution Strategy solutions above; all AFGROW modifications have been summarized here for completeness. Project partner Lextech, Inc. has performed numerous modifications to the AFGROW code necessary to achieve the goals of this project. The status of these tasks is shown in Table 14.

Table 14. Summary of AFGROW Modifications

Item	AFGROW Modification Task: Allow External-K Component (Class) to	Status*
01	be displayed as an option in the AFGROW Input Model menu / select from a list of available plug-in models	C
02	set a name and description for the component, shown in AFGROW specimen view pane	C
03	pass error messages to AFGROW	C
04	draw two-dimensional geometry to the AFGROW specimen view pane	C
05	draw straight and curved crack fronts to the AFGROW specimen view pane	C
06	draw holes and arcs to the AFGROW specimen view pane	C
07	handle AFGROW requests for beta values	C
08	handle AFGROW requests for net section area and cracked section area computations	I**
09	set reference pictures inside AFGROW (static bitmaps) and allow ability to view reference picture during user parameter input (w/o resetting parameters)	C
10	set a list of specimen property names inside AFGROW (each property name has a value)	C
11	set and receive corresponding specimen property values inside AFGROW	C
12	set enumerated property value lists inside AFGROW (e.g., the property value could equal only "fixed", "free", or "pinned" for property name "Right Edge Boundary Condition")	C
13	set a dynamic-length list of property names and values (e.g., as a crack transitions from a through crack to a part-through crack, the property list would add a property name "crack length a" to the existing property name "crack length c")	D
14	set up single or multiple crack fronts inside AFGROW so AFGROW can handle crack propagation	C
15	receive events from AFGROW when the AFGROW user starts or stops a prediction	C
16	receive event from AFGROW when the crack fails or prediction stops due to Kmax exceedance	C

17	receive event from AFGROW when the crack fails or prediction stops due to maximum crack length exceedance	C
18	receive event from AFGROW when the crack fails or prediction stops due to maximum cycle exceedance	C
19	receive event from AFGROW when the crack fails or prediction stops due to user-defined Kmax exceedance	C
20	receive event from AFGROW when the crack fails or prediction stops due to maximum spectrum repetition exceedance	D
21	receive event from AFGROW when the crack fails or prediction stops due to minimum crack growth threshold per spectrum pass not reached	D
22	receive event from AFGROW when the crack fails or prediction stops due to net section yield exceedance (dependent on the external-K component setting net section parameters inside AFGROW)	I**
23	receive event from AFGROW when the crack fails or prediction stops due to crack touched free edge (dependent on the external-K component setting maximum crack length parameters inside AFGROW)	C
24	Provide convergence information to the AFGROW user (via access to AFGROW status bar? Status bar access also req'd during long SC runs)	C
25	Bug Fix: Crack Propagation in second dimension not working	C
26	Bug Fix: Crack Length_limit in second dimension not working	C
27	Bug Fix: Drawing Arcs leaves an extra line in some cases	I
28	Modification: Arcs should not cause auto-scale in AFGROW drawing window	I
29	Bug Fix: Allow Item/Value separator in Specimen Properties Frame to be moved	D
30	Allow multiple crack objects (more than two)	C
31	File Save and Read ability (or allow each class to read/save files?)	C
	* = In-Work; C = Completed; D = Dropped (not needed or cost/time prohibitive)	
	**Dropped by mutual agreement due to difficulty in implementing	

4 Conclusions and Recommendations

4.1 Conclusions

This study has provided the opportunity to construct practical applications for crack growth analysis using an External K-solver tightly integrated with the AFGROW software. The plugin demonstrations illustrate that key technologies have matured enough for implementation into current damage tolerance design tools. The computation requirements normally do not necessitate machine capability beyond the reach of most users, but the present speeds and space requirements in many applications will require patience and/or tabulated solutions for practical reasons. Due to the infancy of the technologies, the initial applications are expected to encounter obstacles, but in our opinion most will have simple resolutions with cooperation from the developers of the K-solver and the developers of AFGROW. At this stage it is advised not to have expectations that the casual damage tolerance analyst can construct and execute interactive K-solver / crack growth models. These early stages of applications will benefit from an integrated group of experts pursuing specific problem types that are of immediate benefit, while also constructing the appropriate infrastructure to further capture the advantages of interactive solutions. This document advocates that the AFRL develop a multi-phased plan that starts with feasible solutions which produce immediate benefit to aircraft structure, then begins to build libraries of simple solutions, and continues on to meet successively more complicated structural problem classes as lessons are learned and processes formulated.

The AFGROW crack growth analysis software has developed a large and significant user base; its platform has achieved levels of confidence and credibility within the aircraft community. Damage tolerance design tools must be organized, understood, accessible, and utilized for the benefits to be fully realized by the USAF and the aircraft industry. The future evolution of integrated External K-solver—AFGROW software into such a tool must take into account specific user design systems requirements to continue to build and maintain confidence and reliability. To demonstrate that integrated External K-solver—AFGROW software can meet those user needs, a major aircraft manufacturer (the Boeing Company) has assisted in defining these system requirements. The maximized Return on Investment (ROI) for the technology resides in the tools being applied to aircraft structure in all phases: from design, through production, into service, and finally into the fleet retirement. Those applications must ensure safety, maximize readiness, and minimize cost.

4.2 Recommendations

Building upon the external-K solver plugin functionality described and demonstrated in this document, additional problem types and classes critical to industry should continue to be identified and implemented within the existing framework of AFGROW and StressCheck®. Using industry guidance to determine the types and the required fidelity of future models is essential for building user acceptance of the solutions.

Windows Terminal Services, the selected method of deploying the beta AFGROW software, the plugin DLL files, and the external-K solver to the user community, was ideal for use as the software capabilities began to mature, software versions changed often, and frequent modifications needed to be made (by separate developers!) to all three portions of the software (AFGROW, the plugin ActiveX DLL's, and StressCheck®). However, as the maturation process of this combined software reaches later stages, it is recommended that the integrated software be disassociated with Windows Terminal Services. For the general user that does not possess his/her own StressCheck® license, the plugins can be modified such that they are intended to be installed locally on client machines along with AFGROW, and only use internet connectivity (http) for passing parameters to the external-K server, then receiving only K results from the server. This will help avoid the multithreading issues with AFGROW that are likely to arise with the continued use of Windows Terminal Services. Furthermore, several large companies and organizations block by default the computer port needed to run Windows Terminal Services using software or hardware firewalls. Users in these organizations must request that this port be opened, which can often be a bureaucratic hassle.

It is also essential that discussions continue between AFRL, APES, Boeing, and ESRD regarding StressCheck® licensing issues. Currently, ESRD is allowing any external-K solver user to access specific solutions using the license purchased by AFRL via APES, but this agreement is only in effect while the integrated software is still in a 'prototype' mode. As the integrated software matures, we will need to jointly evaluate the StressCheck® licensing issues for the general community of external-K solver users.

5 References

[1] APES, Inc., 2002. Crack Growth and Stress Intensity Prediction Techniques, Volume 2: Feasibility Study for Interacting AFGROW With External K-Solvers. Final Report for Contract F33615-98-D-3210 Delivery Order No. 27-3 S1126, June 2002.

[2] Szabó, B. A. and Babuška, I. 1991. Finite Element Analysis, John Wiley and Sons, Inc. New York.

[3] Users Manual for StressCheck®, Version 6.0, Engineering Software Research and Development (ESRD), Inc., 10845 Olive Blvd. Suite 170, St. Louis, Missouri, 63141.

[4] Roark, R.J. and Young, W.C. 1975. Formulas for Stress and Strain, Fifth Edition, McGraw-Hill Book Company. New York.

6   Appendices

6.1 Model Family Documentation: Continuing Damage at a Fastener Hole in an Integrally Stiffened Panel

6.2 Model Family Documentation: Two-Bay Crack in an Integrally Stiffened Panel

6.3 Model Family Documentation: Through Crack at a Semicircular Notch

6.4 Model Family Documentation: Through Crack at a U-shaped Notch

6.5 Model Family Documentation: Through Crack at a Slotted Notch

6.6 Model Family Documentation: Through Crack at a V-Shaped Notch

6.7 Model Family Documentation: Through Crack at a Plate Nut

6.8 Model Family Documentation: Part-Through Crack at a Lug

6.9 Model Family Documentation: Multi-site Damage

6.1 Model Family Documentation: Continuing Damage at a Fastener Hole in an Integrally Stiffened Panel

6.1.1 Introduction

The goal of this family of StressCheck models is to provide pre-meshed models for use with the "Continuing Damage in an Integrally Stiffened Panel" plug-in module (external-K component) for AFGROW. These models are defined in accordance with the geometric limitations specified herein, and are only used during the initial stage (Stage 3, small crack sizes, part-through cracks) of crack propagation analyses. During later cracking stages (Stages 4-6), auto-meshed, extruded StressCheck models are used. All finite element models are three-dimensional models subjected to a far-field tension stress.

6.1.2 Software Version

The StressCheck models use StressCheck Version 6.2.2k.

6.1.3 Sketch of Model Parameters (AFGROW definitions)1.3

This section provides a sketch of the AFGROW user interface. Included in the sketch are the parameter names exposed to the AFGROW interface. AFGROW Beta Version was used.

The following models show the crack front parameter sketch stage descriptions defined in AFGROW:

Crack Stage 3

Crack Stage 4

Crack Stage 5

Crack Stage 6
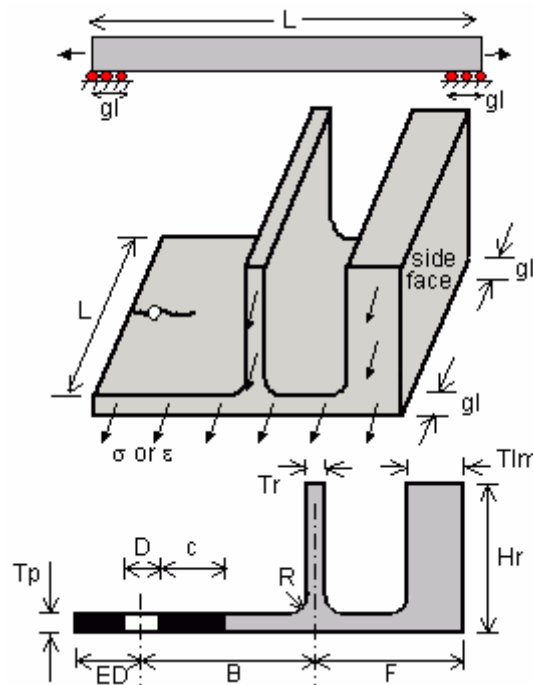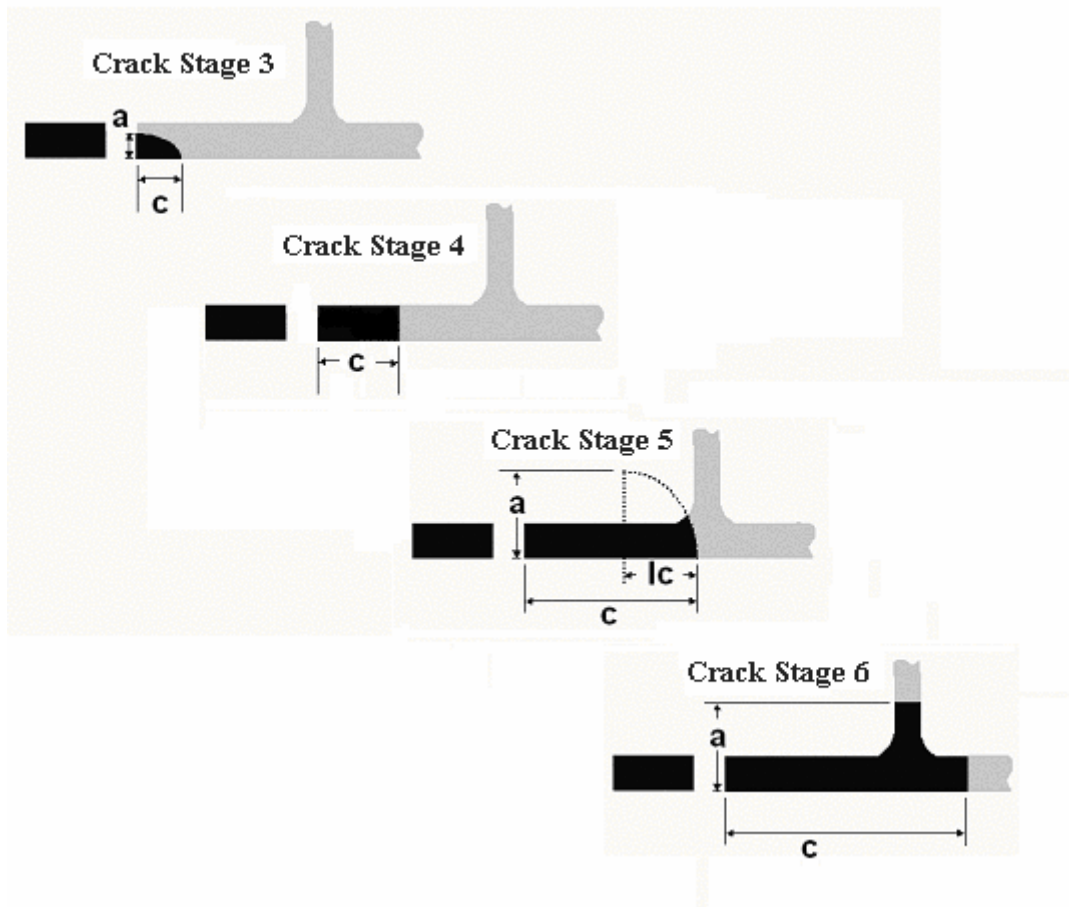
6.1.4 Model Scale

Scaling the finite-element model is often necessary to avoid inherent numerical difficulties with very small numbers. The AFGROW geometric parameter values and limits are multiplied by the model scale before they are used by StressCheck, and the resulting StressCheck Stress Intensity Factors are divided by the square root of the model scale, then used to calculate beta factors for use in AFGROW.

Parameter Ratio	Model Scale
$0.005 \leq a/tp \leq 0.10$	100.0
$0.10 < a/tp \leq 0.90$	10.0

6.1.5 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second

section defines parameters used only by the finite-element models.

6.1.5.1 Parameters in the User Interface (Length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
a	a	crack bore length	0.005	0.90*Tp
B	B	distance of stiffener from hole	3*Tp+0.5*D+R	10.0
c	c	crack surface length	0.5*a	2*a
D	Dh	hole diameter	0.1875	0.5
ED	ED	distance of hole from near edge	1.5*D	B
F	F	distance of stiffener from edge away from hole	0.5*Tr+2*R+tlm+0.03	0.5*Tr+2*R+tlm+20.0
gl	gl	grip length	0	L/2
hlm	hlm	lumped mass height	Tp+R+0.03	2*Hr
Hr	hr	stiffener height	Tp+R+0.25	Tp+R+5.00
L	L	panel half-length	B	4*(ED+B+F)
Modulus	Emod	Young's modulus	no limit	no limit
Nu	v	Poisson's ratio	≥ 0	≤ 0.5
R	Rf	fillet radius	0.125	0.5
Tlm	tlm	lumped mass thickness	Tr	determined by F limits
Tp	tp	panel thickness	0.1	0.5
Tr	tr	stiffener thickness	0.1	0.5

6.1.5.2 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Detail Description
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
rcyl	cylinder radius	Model construction parameter.
St	stress	Applied stress, ksi. Fixed at 1. Stress intensity results from the model are linearly scaled for various applied stress levels.
w	width	Model width. Fixed at ED+B+F.

6.1.6 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
Int_3a1a.sci Int_3a2a.sci	12/8/2004	Small a/t ratio
Int_3b1a.sci Int_3b2a.sci	12/8/2004	Small to medium a/t ratio
Int_3c1a.sci Int_3c2a.sci	12/8/2004	Medium to large a/t ratio

Valid crack length and parameter ranges for each file:

*.sci File Name	Crack/Stiffener Ratio (a/Tp)	Stiffener Distance from Hole (B)
Int_3a1a.sci	$0.005 \leq a/Tp \leq 0.10$	$B < 1.5$
Int_3a2a.sci	$0.005 \leq a/Tp \leq 0.10$	$1.5 \leq B \leq 10.0$
Int_3b1a.sci	$0.10 < a/Tp \leq 0.48$	$B < 2.0$
Int_3b2a.sci	$0.10 < a/Tp \leq 0.48$	$2.0 \leq B \leq 10.0$
Int_3c1a.sci	$0.48 < a/Tp \leq 0.90$	$B < 2.0$
Int_3c2a.sci	$0.48 < a/Tp \leq 0.90$	$2.0 \leq B \leq 10.0$

6.1.7 Sample Snapshots of Finite Element Model(s)

Model Int_3c1a.sci is shown below depicting the stress, symmetry and nodal constraints.
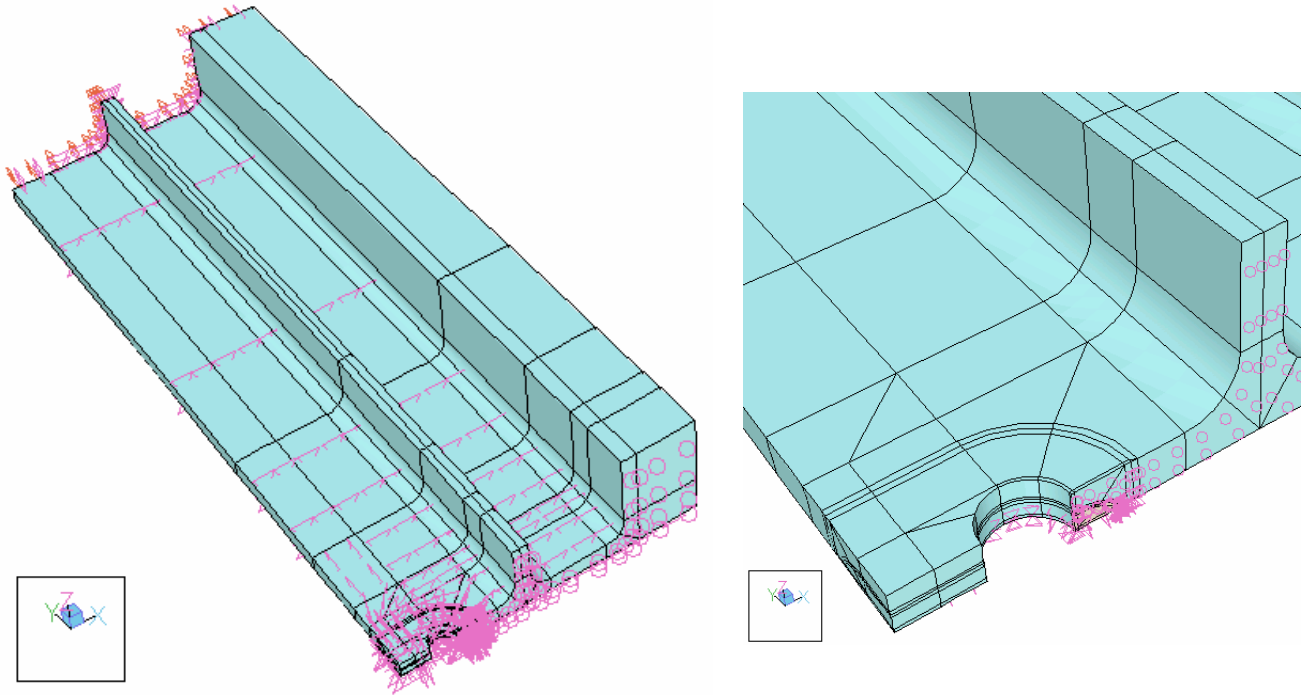
6.1.8 Visual Basic Decision Tree for Selecting Appropriate Model

This documention includes a decision tree to decide which finite element model in the family will be most appropriate. The primary goals are that the selected finite element model is (a) valid, i.e. no element collisions or illegally distorted elements, and (b) gives excellent convergence from p=6 to p=8. "Excellent" convergence is defined here as less than a 1% difference between the p=8 result and an estimated limit. The estimated limit may be either calculated via StressCheck's internal method, or externally via a Richardson extrapolation. The results obtained for the ratio $0.05 \leq a/tp \leq 0.90$ yield excellent convergence. However, the convergence standard is not met for approximately 50% of the cases tested for the very small ratio $0.005 \leq a/tp < 0.01$ at B values approaching the upper limits.

Following is an example of the Visual Basic code used to select the appropriate StressCheck model.

```
Function getModelNameIntStruc(a As Double, c As Double, B As Double, D As
Double, ED As Double, Rf As Double, tp As Double, tr As Double) As String
'selects the sci file to use based on the current parameter combination
Dim tol As Double
tol = 0.00001
```

```
'a series
Select Case (a / tp)
    Case 0.005 To 0.01  'a/tp
        Select Case a
            Case 0.05 To 1.0
                If 0.5 * D + 12 * (a + c) < B - Rf - 0.5 * tr And
                    B < 150 Then
                    getModelNameIntStruc = "Int_3a1.sci"
                ElseIf 0.5 * D + 12 * (a + c) > B - Rf - 0.5 * tr Then
                    getModelNameIntStruc = "ERROR: crack elements
                        exceed B input; B too small" 'no file selected
                Else
                    getModelNameIntStruc = "Int_3a2.sci"   'for B >= 150
                End If
            Case Else
                getModelNameIntStruc = "Invalid: Check input values"
        End Select
'b series
    Case (0.10 + tol) To 0.48    'a/tp
        Select Case a
            Case 0.10 To 2.4
                If 1.5 * (0.5 * D + 0.8 * (a + c)) < B - Rf - 0.5 * tr
                    And B <= 20 Then
                    getModelNameIntStruc = "Int_3b1.sci"
                ElseIf 1.5 * (0.5 * D + 0.8 * (a + c)) >
                    B - Rf - 0.5 * tr Then
                    getModelNameIntStruc = "ERROR: crack elements
                        exceed B input; B too small" 'no file selected
                Else
                    getModelNameIntStruc = "Int_3b2.sci"     'for B > 20
                End If
            Case Else
                getModelNameIntStruc = "Invalid: Check input values"
        End Select
'c series
    Case (0.48 + tol) To 0.8     'a/tp
        Select Case a
            Case 0.48 To 4.0
                If 0.5 * D + 0.8 * (a + c) + 2 * 0.15 * c <
                    B - Rf - 0.5 * tr And B <= 20 Then
                    getModelNameIntStruc = "Int_3c1.sci"
                ElseIf 0.5 * D + 0.8 * (a + c) + 2 * 0.15 * c >
                    B - Rf - 0.5 * tr Then
                    getModelNameIntStruc = "ERROR: crack elements
                        exceed B input; B too small"  'no file selected
                Else
                    getModelNameIntStruc = "Int_3c2.sci"    'for B > 20
                End If
            Case Else
                getModelNameIntStruc = "Invalid: Check input values"
        End Select
    Case (0.8 + tol) To 0.9      'a/tp
        Select Case a
            Case 0.8 To 4.5
                If 0.5 * D + 0.8 * (a + c) + 2 * 0.05 * c <
```

```
                    B - Rf - 0.5 * tr And B <= 20 Then
                getModelNameIntStruc = "Int_3c1.sci"
            ElseIf 0.5 * D + 0.8 * (a + c) + 2 * 0.05 * c >
                    B - Rf - 0.5 * tr Then
                getModelNameIntStruc = "ERROR: crack elements
                    exceed B input; B too small"  'no file selected
            Else
                getModelNameIntStruc = "Int_3c2.sci"    'for B > 20
            End If
        Case Else
            getModelNameIntStruc = "Invalid: Check input values"
        End Select
    End Select
End Function
```

6.2   Model Family Documentation: Two-Bay Crack in an Integrally Stiffened Panel

6.2.1 Introduction

The goal of this family of StressCheck models is to provide models for use with the "Two Bay Crack in an Integrally Stiffened Panel" plug-in module (external-K component) for AFGROW. The models are defined in accordance with the geometric limitations specified herein, and are used during the later cracking stages (Stages 4-6) for auto-meshed, extruded StressCheck models. All finite element models are three-dimensional models subjected to a far-field tension stress.

6.2.2 Software Version

The StressCheck models use StressCheck Version 6.2.2k.

6.2.3 Sketch of Model Parameters (AFGROW definitions)

This section provides a sketch of the AFGROW user interface. Included in the sketch are the parameter names exposed to the AFGROW interface. AFGROW Beta Version was used.
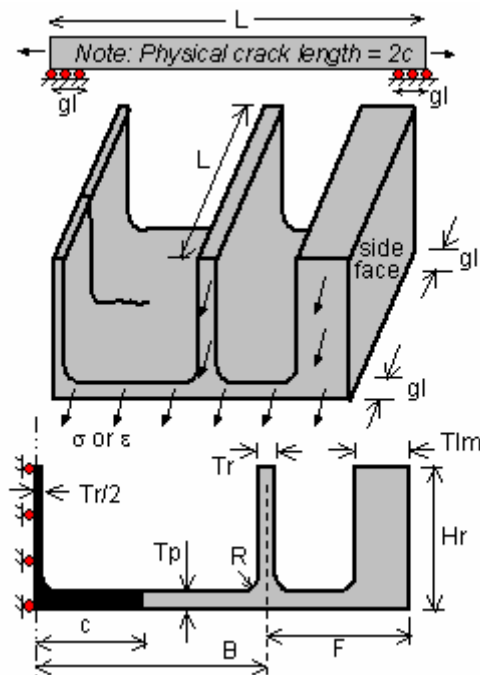
The following models show the crack front parameter sketch stage descriptions defined in AFGROW:
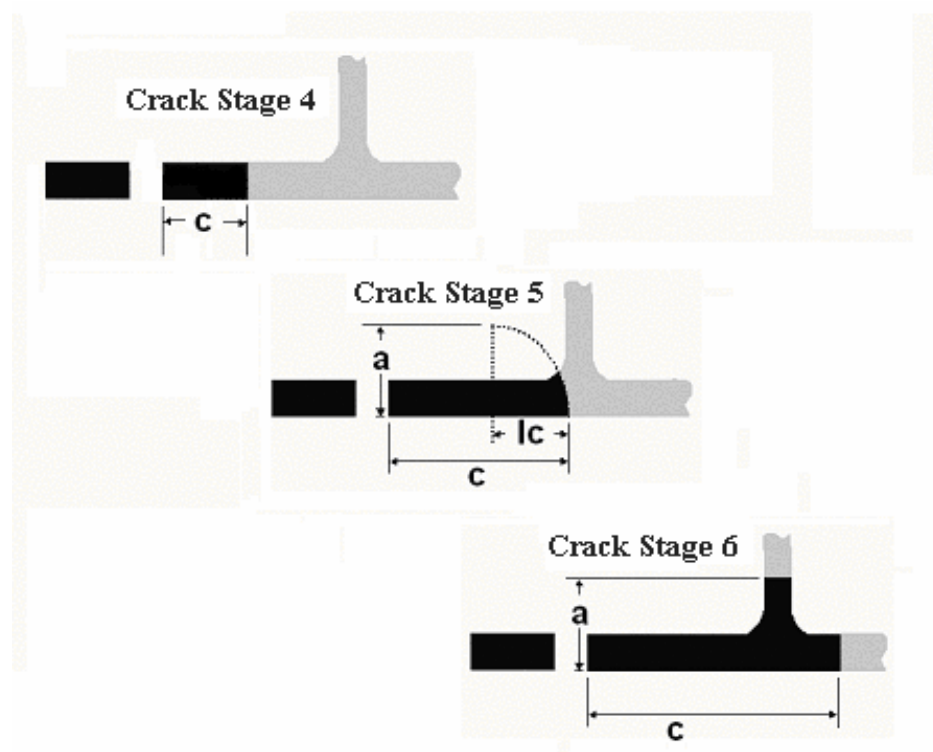
6.2.4 Parameter Names, Definitions, and Limits

This section contains the parameter names related to the AFGROW interface.

Parameters in the User Interface (Length dimensions in inches)

AFGROW Parameter	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
a	crack bore length	> Tp	< Hr
B	distance of stiffener from hole	3*Tp+0.5*D+R	10.0
c	half-crack length	Tr/2+R+0.03	B+F-Tlm-R-0.03
F	distance of stiffener from edge away from hole	0.5*Tr+2*R+tlm+0.03	0.5*Tr+2*R+tlm+20.0
gl	grip length	0	L/2
Hr	stiffener height	Tp+R+0.25	Tp+R+5.00
L	panel length	B	4*(ED+B+F)
lc	local ellipse axis (Stage 5)	0	C
Modulus	Young's modulus	no limit	no limit
Nu	Poisson's ratio	≥ 0	≤ 0.5
R	fillet radius	0.125	0.5
Tlm	lumped mass thickness	Tr	determined by F limits
Tp	panel thickness	0.1	0.5
Tr	stiffener thickness	0.1	0.5

6.3   Model Family Documentation: Through Crack at a Semicircular Notch

6.3.1 Introduction

The goal of this family of StressCheck models is to provide an external K-solver component for AFGROW for a Through Crack at a Semicircular Notch of almost arbitrary radius. This problem family differs from the existing AFGROW solution in that it allows an arbitrary notch radius to be specified, in contrast with the current version of AFGROW, which only allows a notch radius equal to 0.0625 times the part width. The only load condition allowed is a far-field tensile stress. All finite element models are two-dimensional.

6.3.2 Sketch of Model Parameters (AFGROW definitions)

This section provides sample sketches that an AFGROW user might see. Included in the sketch are the parameter names exposed to the AFGROW interface.
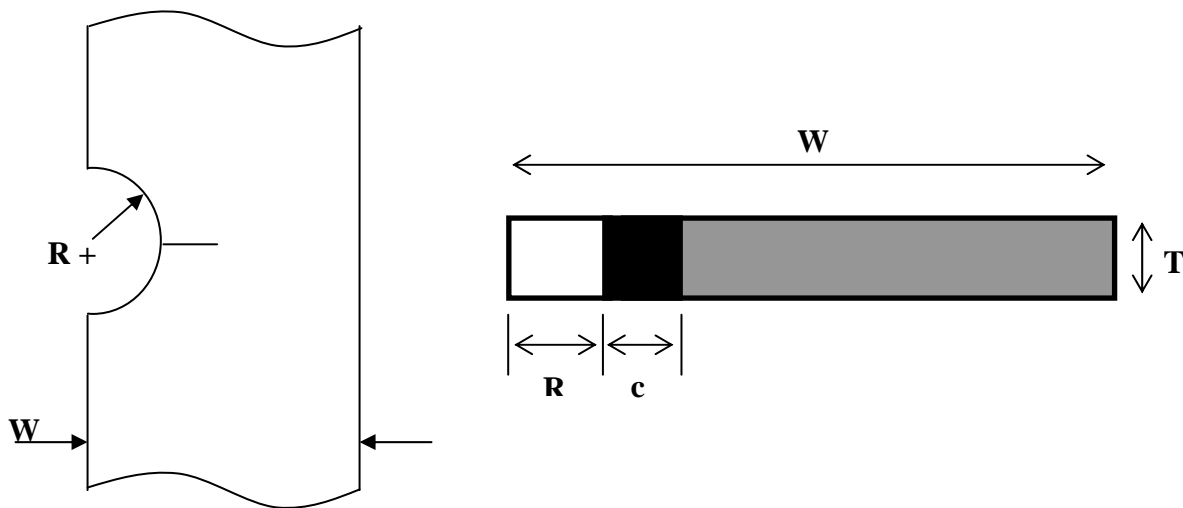
6.3.3 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

6.3.3.1 Parameters in the User Interface (Length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
W	w	width	1	10
R	rd	notch radius	0.10*W	0.80*W
c	a	crack length	0.005*W	0.80*(W-R)
T	th	thickness	no limit	no limit

6.3.3.2 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Detail Description
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
h	height	Height of the half-model. h is fixed at 2*width to avoid finite-length plate influence of a near-field applied stress.
St	stress	Applied stress, ksi. Fixed at 1. Stress intensity results from the model are linearly scaled for various applied stress levels.

6.3.4 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
notchA2.sci	05.14.2004	Small radii, small cracks
notchC6.sci	05.14.2004	Small to medium radii, small cracks
notchB3.sci	05.14.2004	Medium to large radii, small cracks
notchD1.sci	05.14.2004	Small to medium radii, large cracks
notchD4.sci	05.14.2004	Medium to large radii, large cracks

Valid crack length and notch radius parameter ranges for each file:

Crack Range (c/W)	Notch Radius (R)	*.sci File Name
$0.005 \leq c/W < 0.100$	$0.0625W \leq R < 0.10W$	notchA2.sci
$0.005 \leq c/W < 0.100$	$0.10W \leq R < 0.40W$	notchC6.sci
$0.005 \leq c/W < 0.100$	$0.40W \leq R \leq 0.80W$	notchB3.sci
$0.100 \leq c/W \leq 0.80(W-R)$	$0.0625W \leq R < 0.40W$	notchD1.sci
$0.100 \leq c/W \leq 0.80(W-R)$	$0.40W \leq R \leq 0.80W$	notchD4.sci

6.3.5 Sample Snapshots of Finite Element Model(s)

Model notchA2.sci is shown in two different views: the left figure depicts the full model with the input stress field, symmetry and nodal constraints, while the right figure depicts mesh detail near the crack tip.

6.3.6 Visual Basic Decision Tree for Selecting Appropriate Model A set of parameters is input to the external-K component, and a decision tree is necessary to decide which finite element model in the family will be most appropriate. The primary goals are that the selected finite element model is (a) valid, i.e. no element collisions or illegally distorted elements, and (b) gives excellent convergence from p=6 to p=8. "Excellent" convergence is defined here as less than a 1% difference between the p=8 result and an estimated limit. The estimated limit may be either calculated via StressCheck's internal method, or externally via a Richardson extrapolation. However, it is not expected that the selected model for *any* parameter combination within the allowable ranges will provide excellent convergence.

Following is an example of the Visual Basic code used to select the appropriate StressCheck model.

```
Function getModelNameSCN(a As Double, r As Double, w As Double) As String
'selects the sci file to use based on the current parameter combination
Dim amax As Double, tol As Double
tol = 0.00001   'tolerance
amax = 0.8 * (w - r)+ tol
    Select Case r      'notch radius
        Case 0.0625 * w To (0.1 * w - tol)
            Select Case (a / w) 'crack length/width ratio
                Case 0.005 To (0.1 - tol)
                    getModelNameSCN = "notchA2.sci"
                Case 0.1 To amax
                    getModelNameSCN = "notchD1.sci"
                Case Else
                    getModelNameSCN = "invalid value of a"
            End Select  'a/w
        Case 0.1 * w To (0.4 * w - tol) 'notch radius
            Select Case (a / w)
                Case 0.005 To (0.1 - tol)
                    getModelNameSCN = "notchC6.sci"
                Case 0.1 To amax
                    getModelNameSCN = "notchD1.sci"
```

```
            Case Else
                getModelNameSCN = "invalid value of a"
        End Select  'a/w
    Case 0.4 * w To 0.8 * w  'notch radius
        Select Case (a / w)
            Case 0.005 To (0.1 - tol)
                getModelNameSCN = "notchB3.sci"
            Case 0.1 To amax
                getModelNameSCN = "notchD4.sci"
            Case Else
                getModelNameSCN = "invalid value of a"
        End Select  'a/w
    Case Else    'notch radius
        getModelNameSCN = "invalid value of notch radius"
  End Select
```

6.3.7 Verification of FE Models: Checks with Industry Solutions (where available)

6.3.7.1 Sample results for model *notchA2.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter Value
a	c	0.02 in.	0.02 in.
dim	N/A	0.15	N/A
h	N/A	2 in.	N/A
rd	R	0.0625 in.	0.0625 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 0.5755 ksi-in$^{1/2}$ and β = 2.294. The StressCheck computed Limit K1 is 0.5751 ksi-in$^{1/2}$.

For the AFGROW specific case of R=0.0625W, the AFGROW computed Stress Intensity Factor is K = 0.5788 ksi-in$^{1/2}$ and β = 2.309. This is less than one percent difference from the finite element computed K of 0.5751 ksi-in$^{1/2}$.

6.3.7.2 Sample results for model *notchD1.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter value
a	c	0.538 in.	0.538 in.
dim	N/A	0.15	N/A
h	N/A	2 in.	N/A
rd	R	0.0625 in.	0.0625 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 5.549 ksi-in$^{1/2}$ and β = 4.268. The StressCheck computed Limit K1 of 5.549 ksi-in$^{1/2}$.

For the AFGROW specific case of R=0.0625W, the AFGROW computed Stress Intensity Factor is K = 5.442 and β = 4.186. This is a 1.97 percent difference from the finite element computed K of 5.549 ksi-in$^{1/2}$.

6.3.8 Sample data using FE Models

Sample results for various models and notch configurations. StressCheck version V6.3.1f (Beta) was used. For all input h = 2, W = 1, and St = 1. Integration radius = 0.075*a.

*.sci Parameter Name	Crack Length a	Notch Radius R	Stress Intensity K1	β
notchA2.sci	0.005	0.0625	3.827e-01	3.054
notchA2.sci	0.100	0.100	1.084e+00	1.934
notchC6.sci	0.005	0.200	5.219e-01	4.164
notchC6.sci	0.100	0.300	2.306e+00	4.114
notchB3.sci	0.005	0.400	9.205e-01	7.345
notchB3.sci	0.100	0.500	5.312e+00	9.477
notchD1.sci	0.500	0.100	5.536e+00	4.417
notchD1.sci	0.400	0.250	7.121e+00	6.352
notchD4.sci	0.300	0.500	1.896e+01	19.530
notchD4.sci	0.100	0.750	3.025e+01	53.970

6.4   Model Family Documentation: Through Crack at a U-shaped Notch

6.4.1 Introduction

The goal of this family of StressCheck models is to provide an external K-solver component for AFGROW for a Through Crack at a Shallow U-Shaped Notch. These problem models are defined by the plate dimensions, notch radius and notch width. The only load condition allowed is a far-field tensile stress. All finite element models are two-dimensional.

6.4.2 Sketch of Model Parameters (AFGROW definitions)

This section provides sample sketches that an AFGROW user might see. Included in the sketch are the parameter names exposed to the AFGROW interface.
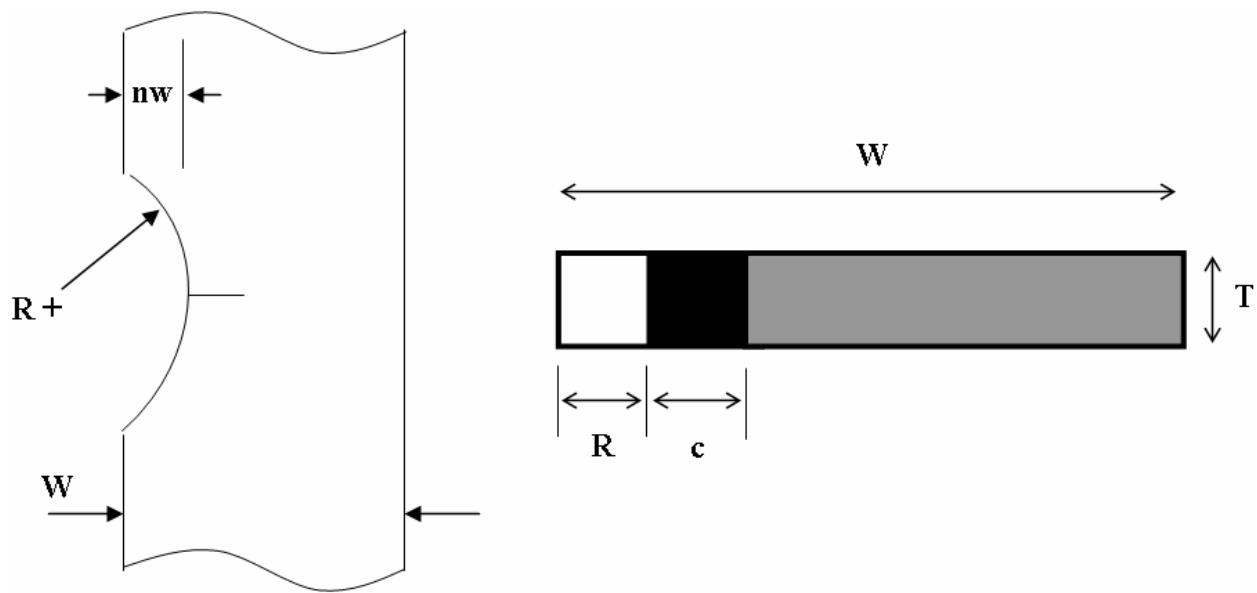
6.4.3 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

6.4.3.1 Parameters in the User Interface (All length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
W	w	width	1	10
R	rd	notch radius	1.105*nw	$(4*W+nw^2)/(2*nw)$
c	a	crack length	0.005*W	0.80*(W-nw)
nw	nw	notch width	0.10*W	0.80*W
T	th	thickness	no limit	no limit

6.4.3.2 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Detail Description
cd	r center location	Location of the notch radius center.
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
h	height	Height of the half-model. h is fixed at 4*width to avoid finite-length plate influence of a near-field applied stress.
nh	notch height	Model construction parameter. nh is a function of the notch radius and notch width.
St	stress	Applied stress, ksi. Fixed at 1. Stress intensity results from the model are linearly scaled for various applied stress levels.

6.4.4 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
shallow U-notchA3.sci	05.14.2004	Small to medium radii, small cracks
shallow U-notchB3.sci	05.14.2004	Medium to large radii, small cracks
shallow U-notchF3.sci	05.14.2004	Small to medium radii, large cracks
shallow U-notchF4.sci	05.14.2004	Medium to large radii, large cracks

Valid crack length and notch radius parameter ranges for each file:

Crack Range (a)	Notch Width (nw)	*.sci File Name
0.005 ≤ a/W < 0.100	0.10W ≤ nw < 0.40W	shallow U-notchA3.sci
0.005 ≤ a/W < 0.100	0.40W ≤ nw < 0.80W	shallow U-notchB3.sci
0.10 ≤ a/W ≤ 0.80(W-nw)	0.10W ≤ nw ≤ 0.40W	shallowU-notchF3.sci
0.10 ≤ a/W ≤ 0.80(W-nw)	0.40W ≤ nw ≤ 0.80W	shallowU-notchF4.sci
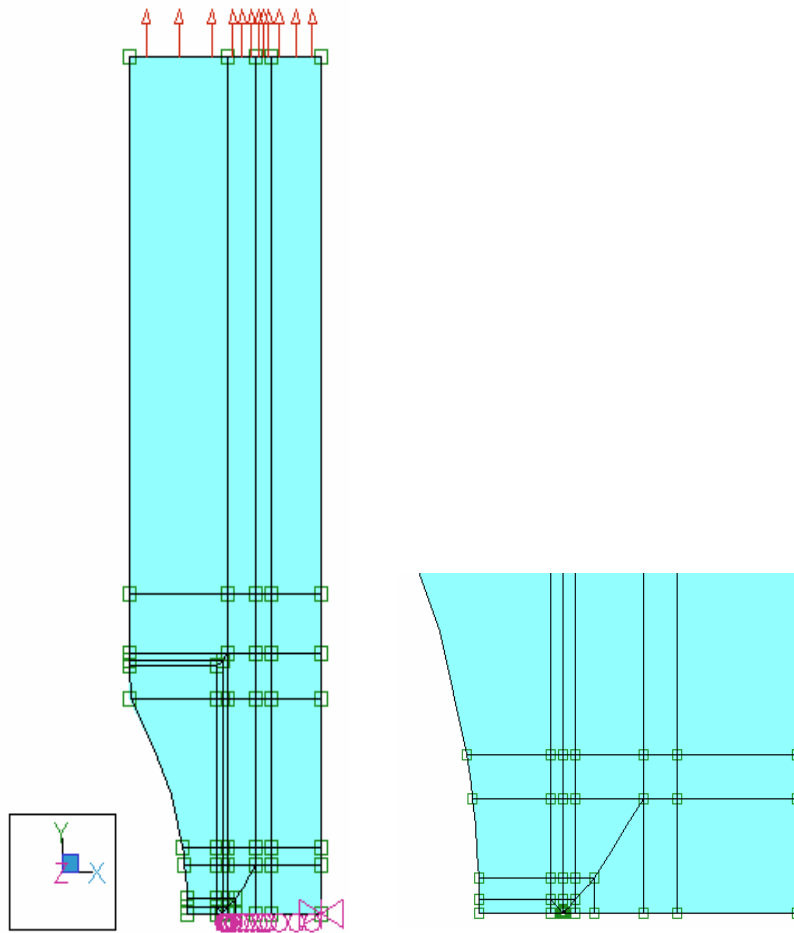
6.4.5 Sample Snapshots of Finite Element Model(s)

Model shallowU-notchA3.sci is shown in two different views: the left figure depicts the full model with the input stress field, symmetry and nodal constraints, while the right figure depicts mesh detail near the crack tip.

6.4.6 Visual Basic Decision Tree for Selecting Appropriate Model

A set of parameters is input to the external-K component, and a decision tree is necessary to decide which finite element model in the family will be most appropriate. The primary goals are that the selected finite element model is (a) valid, i.e., no element collisions or illegally distorted elements, and (b) gives excellent convergence from p=6 to p=8. "Excellent" convergence is defined here as less than a 1% difference between the p=8 result and an estimated limit. The estimated limit may be either calculated via StressCheck's internal method, or externally via a Richardson extrapolation. However, it is not expected that the selected model for *any* parameter combination within the allowable ranges will provide excellent convergence.

Following is an example of the Visual Basic code used to select the appropriate StressCheck model.

```
Function getModelNameShallowU_Notch(a As Double, r As Double, nw As Double, w
As Double) As String
'selects the sci file to use based on the current parameter combination
Dim amax As Double, Rmin As Double, Rmax As Double, tol As Double
tol = 0.00001    'tolerance
amax = 0.8 * (w - nw) + tol
nwmax = 0.8 * w + tol
Rmin = 1.105 * nw
Rmax = (4 * w + nw ^ 2) / (2 * nw) + tol
'nh = Sqr(2 * rd * nw - nw ^ 2)
Select Case nw      'notch width
    Case 0.1 * w To (0.4 * w - tol)
        Select Case r    'notch radius
            Case Rmin To Rmax
                Select Case (a / w) 'crack length
                    Case 0.005 To (0.1 - tol)
                        getModelNameShallowU_Notch = "shallowU-notchA3.sci"
                    Case 0.1 To amax
                        getModelNameShallowU_Notch = "shallowU-notchF3.sci"
                    Case Else
                        getModelNameShallowU_Notch = "invalid value of a"
                End Select  'a/w
            Case Else
                getModelNameShallowU_Notch = "invalid value of notch radius"
            End Select  'r
    Case 0.4 * w To nwmax 'notch width
        Select Case r    'notch radius
            Case Rmin To Rmax
                Select Case (a / w) 'crack length
                    Case 0.005 To (0.1 - tol)
                        getModelNameShallowU_Notch = "shallowU-notchB3.sci"
                    Case 0.1 To amax
                        getModelNameShallowU_Notch = "shallowU-notchF4.sci"
                    Case Else
                        getModelNameShallowU_Notch = "invalid value of a"
                End Select  'a/w
            Case Else
                getModelNameShallowU_Notch = "invalid value of notch radius"
            End Select  'r
```

```
    Case Else     'notch width
        getModelNameShallowU_Notch = "invalid value of notch width"
    End Select
End Function
```

6.4.7 Sample Results from FE Model(s)

6.4.7.1 Sample results for model *shallowU-notchA3.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter Value
a	c	0.02 in.	0.02 in.
dim	N/A	0.15	N/A
h	N/A	4 in.	N/A
nw	nw	0.10 in.	0.10 in.
rd	R	0.1105 in.	0.1105 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 0.6719 ksi-in$^{1/2}$ and β = 2.680. The StressCheck computed Limit K1 is 0.6719 ksi-in$^{1/2}$.

6.4.7.2 Sample results for model *shallowU-notchF3.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter Value
a	c	0.50 in.	0.50 in.
dim	N/A	0.15	N/A
h	N/A	4 in.	N/A
nw	nw	0.10 in.	0.10 in.
rd	R	20.00 in.	20.00 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 5.538 ksi-in$^{1/2}$ and β = 4.419. The StressCheck computed Limit K1 is 5.538 ksi-in$^{1/2}$.

6.4.7.3 Sample results for various models and notch configurations. StressCheck version V6.3.1f (Beta) was used. For all input h = 4, W = 1, and St = 1. Integration radius = 0.075*a.

*.sci Parameter Name	Crack Length a	Notch Width nw	Notch Radius R	Stress Intensity K1	β
shallowU-notchA3.sci	0.005	0.10	0.1105	4.087e-01	3.261
shallowU-notchA3.sci	0.100	0.20	10.100	1.417e+00	2.528
shallowU-notchB3.sci	0.005	0.40	0.450	9.035e-01	7.209
shallowU-notchB3.sci	0.100	0.60	0.850	9.077e+00	16.194
shallowU-notchB3.sci	0.005	0.80	2.900	9.055e+00	72.248
shallowU-notchF3.sci	0.100	0.10	5.050	9.835e-01	1.755
shallowU-notchF3.sci	0.500	0.20	3.900	9.423e+00	7.518
shallowU-notchF3.sci	0.300	0.30	1.500	5.523e+00	5.689
shallowU-notchF4.sci	0.200	0.60	3.500	1.895e+01	23.907
shallowU-notchF4.sci	0.100	0.80	2.900	5.821e+01	103.854

6.5 Model Family Documentation: Through Crack at a Slotted Notch

6.5.1 Introduction

The goal of this family of StressCheck models is to provide an external K-solver component for AFGROW for a Through Crack at a Slotted Notch. The problem models are defined by the plate dimensions, slot width and notch radius. The only load condition allowed is a far-field tensile stress. All finite element models are two-dimensional.

6.5.2 Sketch of Model Parameters (AFGROW definitions)

This section provides sample sketches that an AFGROW user might see. Included in the sketch are the parameter names exposed to the AFGROW interface.
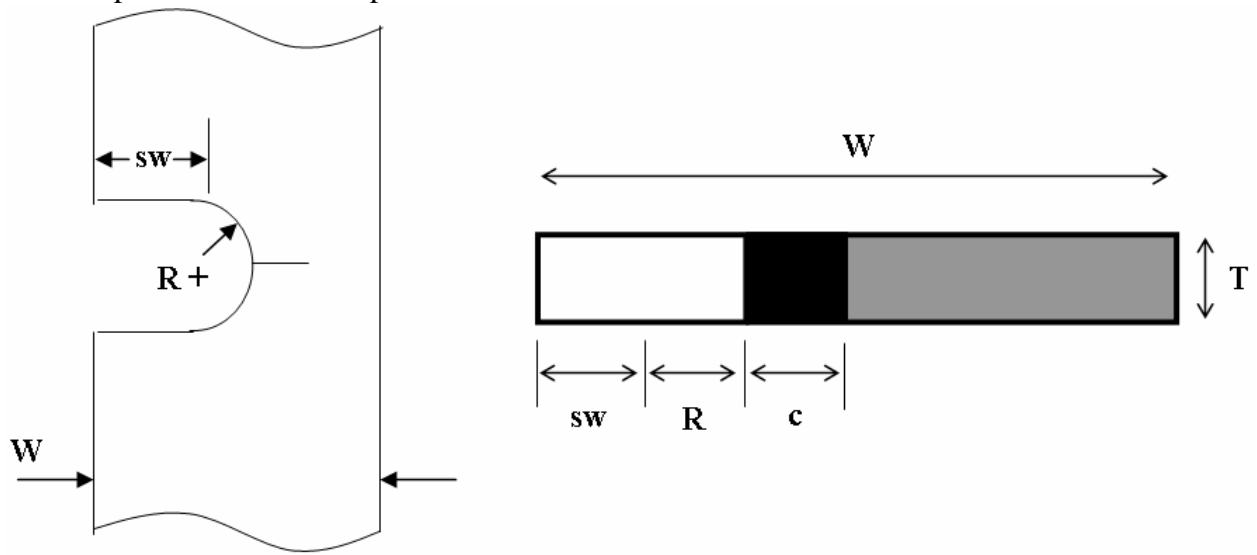
6.5.3 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

6.5.3.1 Parameters in the User Interface (All length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
W	w	width	1	10
R	rd	notch radius	0.0625*W	0.77*W
c	a	crack length	0.005*W	0.80*(W-R-sw)
sw	sw	slot width	0.03*W	0.80*W - R
T	th	thickness	no limit	no limit

Parameters Specific to the Finite-Element Models6.5.3.2

*.sci Parameter Name	*.sci Parameter Description	Detail Description
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
h	height	Height of the half-model. h is fixed at 2*width to avoid finite-length plate influence of a near-field applied stress.
St	stress	Applied stress, ksi. Fixed at 1. Stress intensity results from the model are linearly scaled for various applied stress levels.

6.5.4 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
slotA1.sci	05.14.2004	Small radii, small cracks
slotB1.sci	05.14.2004	Small to medium radii, small cracks
slotC1.sci	05.14.2004	Medium to large radii, small cracks
slotD1.sci	05.14.2004	Small to medium radii, large cracks
slotD2.sci	05.14.2004	Medium to large radii, large cracks

Valid crack length and notch radius parameter ranges for each file:

Crack Range (c/w)	Notch Radius (R)	Slot Width (sw)	*.sci File Name
0.005 ≤ c/W < 0.100	0.0625W ≤ R < 0.10W	0.030 ≤ sw ≤ 0.80W-R	slotA1.sci
0.005 ≤ c/W < 0.100	0.10W ≤ R < 0.40W	0.030 ≤ sw ≤ 0.80W-R	slotB1.sci
0.005 ≤ c/W < 0.100	0.40W ≤ R ≤ 0.77W	0.030 ≤ sw ≤ 0.80W-R	slotC1.sci
0.100 ≤ c/W ≤ 0.80(W-R-sw)	0.0625W ≤ R < 0.40W	0.030 ≤ sw ≤ 0.80W-R	slotD1.sci
0.100 ≤ c/W ≤ 0.80(W-R-sw)	0.40W ≤ R ≤ 0.77W	0.030 ≤ sw ≤ 0.80W-R	slotD2.sci

6.5.5 Sample Snapshots of Finite Element Model(s)

Model slotB1.sci is shown in two different views: the left figure depicts the full model with the input stress field, symmetry and nodal constraints, while the right figure depicts mesh detail near the crack tip.

6.5.6 Visual Basic Decision Tree for Selecting Appropriate Model

A set of parameters is input to the external-K component, and a decision tree is necessary to decide which finite element model in the family will be most appropriate. The primary goals are that the selected finite element model is (a) valid, i.e., no element collisions or illegally distorted elements, and (b) gives excellent convergence from p=6 to p=8. "Excellent" convergence is defined here as less than a 1% difference between the p=8 result and an estimated limit. The estimated limit may be either calculated via StressCheck's internal method, or externally via a Richardson extrapolation. However, it is not expected that the selected model for *any* parameter combination within the allowable ranges will provide excellent convergence.

Following is an example of the Visual Basic code used to select the appropriate StressCheck model.

```
Function getModelNameSLOT(a As Double, r As Double, sw As Double, w As
Double) As String
'selects the sci file to use based on the current parameter combination
Dim amax As Double, swmax As Double, tol As Double
tol = 0.00001   'tolerance
amax = 0.8 * (w – r – sw) + tol
swmax = 0.8 * w – r + tol
    Select Case r  'notch radius
        Case 0.0625 * w To (0.1 * w - tol)
            Select Case sw  'slot width
                Case (0.03 * w) To swmax    'for all values of sw
```

```
                Select Case (a / w) 'crack length
                    Case 0.005 To (0.1 - tol)
                        getModelNameSLOT = "slotA1.sci"
                    Case 0.1 To amax
                        getModelNameSLOT = "slotD1.sci"
                    Case Else
                        getModelNameSLOT = "invalid value of a"
                End Select  'a/w
            Case Else
                getModelNameSLOT = "invalid value of slot width"
        End Select  'slot width
    Case 0.1 * w To (0.4 * w - tol) 'notch radius
        Select Case sw      'slot width
            Case (0.03 * w) To swmax    'for all values of sw
                Select Case (a / w) 'crack length
                    Case 0.005 To (0.1 - tol)
                        getModelNameSLOT = "slotB1.sci"
                    Case 0.1 To amax
                        getModelNameSLOT = "slotD1.sci"
                    Case Else
                        getModelNameSLOT = "invalid value of a"
                End Select  'a/w
            Case Else
                getModelNameSLOT = "invalid value of slot width"
        End Select  'slot width
    Case 0.4 * w To 0.77 * w 'notch radius
        Select Case sw      'slot width
            Case (0.03 * w) To swmax    'for all values of sw
                Select Case (a / w) 'crack length
                    Case 0.005 To (0.1 - tol)
                        getModelNameSLOT = "slotC1.sci"
                    Case 0.1 To amax
                        getModelNameSLOT = "slotD2.sci"
                    Case Else
                        getModelNameSLOT = "invalid value of a"
                End Select  'a/w
            Case Else
                getModelNameSLOT = "invalid value of slot width"
        End Select  'slot width
    Case Else
        getModelNameSLOT = "invalid value of notch radius"
    End Select  'notch radius
End Function
```

6.5.7 Verification of FE Model(s)

6.5.7.1 Sample results for model *slotA1.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter value
a	c	0.02 in.	0.02 in.
dim	N/A	0.15	N/A
h	N/A	2 in.	N/A
rd	R	0.0625 in.	0.0625 in.
St	SMF	1 ksi	1 ksi
sw	sw	.03 in.	.03 in.
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 0.6841 ksi-in$^{1/2}$ and β = 2.729. The StressCheck computed Limit K1 is 0.6841 ksi-in$^{1/2}$.

6.5.7.2 Sample results for model *slotD1.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCheck Parameter Name	AFGROW Parameter Name	StressCheck Parameter Value	AFGROW Parameter value
a	c	0.50 in.	0.50 in.
dim	N/A	0.15	N/A
h	N/A	2 in.	N/A
rd	R	0.0625 in.	0.0625 in.
St	SMF	1 ksi	1 ksi
sw	sw	.03 in.	.03 in.
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 5.341 ksi-in$^{1/2}$ and β = 4.262. The StressCheck computed Limit K1 is 5.341 ksi-in$^{1/2}$.

6.5.8 Sample data using FE Models

Sample results for various models and notch configurations. StressCheck version V6.3.1f (Beta) was used. For all input h = 2, W = 1, and St = 1. Integration radius = 0.075*a.

*.sci Parameter Name	Crack Length a	Notch Radius R	Slot Width sw	Stress Intensity K1	β
slotA1.sci	0.005	0.0625	0.03	4.546e-01	3.627
slotA1.sci	0.050	0.0625	0.50	5.795e+00	14.622
slotB1.sci	0.005	0.100	0.03	4.799e-01	3.820
slotB1.sci	0.050	0.300	0.40	1.208e+01	30.479
slotC1.sci	0.005	0.500	0.03	1.567e+00	12.503
slotC1.sci	0.050	0.750	0.05	2.902e+01	73.221
slotD1.sci	0.500	0.0625	0.03	5.341e+00	4.262
slotD1.sci	0.150	0.250	0.50	5.829e+00	84.913
slotD2.sci	0.100	0.500	0.03	6.182e+00	11.029
slotD2.sci	0.150	0.750	0.03	1.019e+02	148.441

6.6   Model Family Documentation: Through Crack at a V-Shaped Notch

6.6.1 Introduction

The goal of this family of StressCheck models is to provide an external K-solver component for AFGROW for a Through Crack at a V-Shaped Notch. These problem models are defined by the plate dimensions, notch radius, notch height and notch width. The only load condition allowed is a far-field tensile stress. All finite element models are two-dimensional.

6.6.2 Sketch of Model Parameters (AFGROW definitions)

This section provides sample sketches that an AFGROW user might see. Included in the sketch are the parameter names exposed to the AFGROW interface.
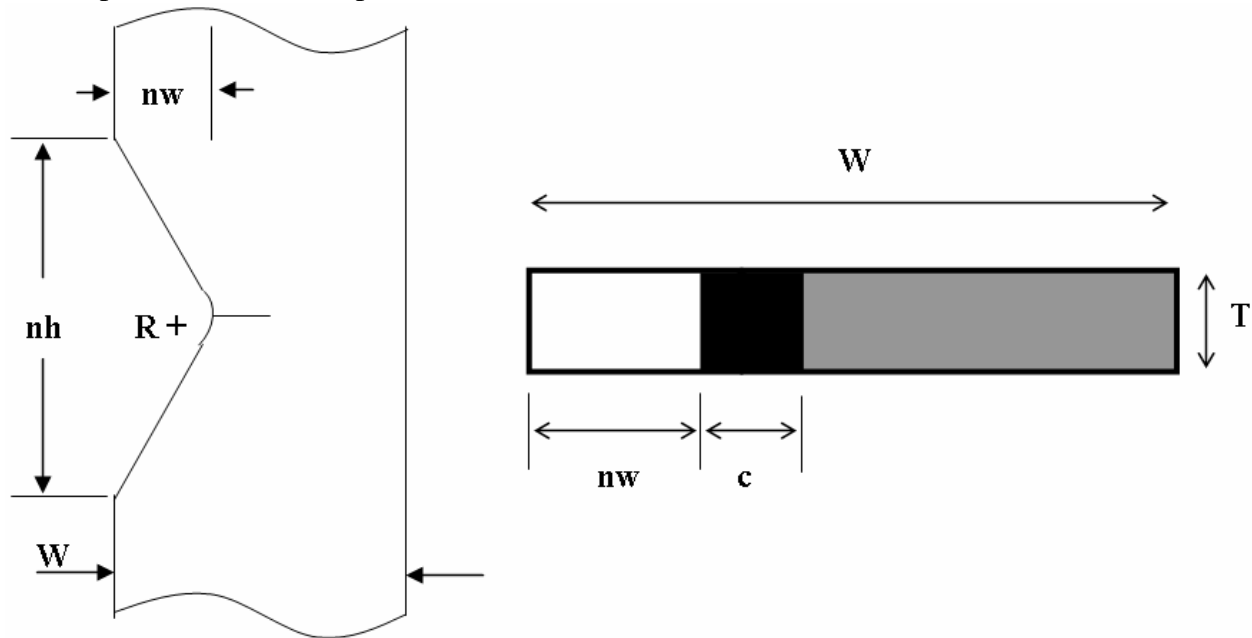
6.6.3 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

6.6.3.2 Parameters in the User Interface (All length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
W	w	width	1	1
R	rd	notch radius	0.10	1.0
c	a	crack length	0.005	0.80(W-nw)
nh	nh	notch height	1.1*R	2
nw	nw	notch width	0.03	0.80
T	th	thickness	no limit	no limit

6.6.3.3 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Detail Description
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
fb	formula value	Model construction parameter used in formulas to control element meshing.
h	height	Height of the half-model. h is fixed at 4*width to avoid finite-length plate influence of a near-field applied stress.
St	stress	Applied stress. Fixed at 1. Stress intensity results from the model are linearly scaled for various applied stress levels.

6.6.4 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
V-notchC4.sci	06.04.2004	Small cracks, minimal to large notch height and width, variable notch radii
V-notchA7.sci	06.04.2004	
V-notchA5.sci	03.26.2004	Small cracks, minimal to moderate notch height, width, and radii
V-notchB1.sci	06.04.2004	
V-notchB2.sci	06.04.2004	Small cracks, moderate to large notch height, width and radii
V-notchB3.sci	06.04.2004	
V-notchC5.sci	06.04.2004	
V-notchE3.sci	06.04.2004	Medium to large cracks, minimal notch radii
V-notchD1.sci	06.04.2004	Medium to large cracks, minimal to moderate notch radii
V-notchD2.sci	06.04.2004	Moderate to large cracks, moderate to large notch radii
V-notchD3.sci		

6.6.5 Sample Snapshots of Finite Element Model(s)

Model V-notchB1.sci is shown in two different views; the left figure depicts the full model with the input stress field, symmetry and nodal constraints, while the right figure depicts crack mesh detail near the crack tip.

85

6.6.6 Visual Basic Decision Tree for Selecting Appropriate Model

A set of parameters is input to the external-K component, and a decision tree is necessary to decide which finite element model in the family will be most appropriate. The primary goals are that the selected finite element model is (a) valid, i.e. no element collisions or illegally distorted elements, and (b) gives excellent convergence from p=6 to p=8. "Excellent" convergence is defined here as less than a 1% difference between the p=8 result and an estimated limit. The estimated limit may be either calculated via StressCheck's internal method, or externally via a Richardson extrapolation. However, it is not expected that the selected model for *any* parameter combination within the allowable ranges will provide excellent convergence.

Following is an example of the Visual Basic code used to select the appropriate StressCheck model.

```
Function getModelNameV_Notch(a As Double, r As Double, nw As Double, nh As
Double) As String
'selects the sci file to use based on the current parameter combination
Dim amax As Double, nwmax As Double, tol As Double, w As Double
w = 1#   'w fixed at 1
tol = 0.00001    'tolerance
amax = 0.8 * (w - nw) + tol
nwmax = 0.8 + tol

Select Case r    'notch radius
    Case 0.1 To (0.2 - tol)
        Select Case a   'crack length
            Case 0.005 To (0.05 - tol)
                Select Case nh   'notch height
                    Case 0.11 To (0.15 - tol)
                        Select Case nw   'notch width
                            Case 0.03 To nwmax
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case Else    'nw
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 0.15 To (0.25 - tol)    'nh
                        Select Case nw
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchA5.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 0.25 To (0.5 - tol) 'nh
                        Select Case nw
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchA7.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 0.5 To (1# - tol)    'nh
                        Select Case nw
                            Case 0.03 To (0.4 - tol)
                                getModelNameV_Notch = "V-notchA7.sci"
                            Case 0.4 To nwmax
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 1# To (2# + tol)   'nh
                        Select Case nw
                            Case 0.03 To (0.4 - tol)
```

```
                              getModelNameV_Notch = "V-notchA7.sci"
                        Case 0.4 To nwmax
                              getModelNameV_Notch = "V-notchC4.sci"
                        Case Else
                              getModelNameV_Notch = "invalid notch width"
                  End Select
            Case Else    'nh
                  getModelNameV_Notch = "invalid notch height"
         End Select
      Case 0.05 To amax    'a
         Select Case nh
            Case (1.1 * r) To (0.25 - tol)
                  Select Case nw
                        Case 0.03 To nwmax
                              getModelNameV_Notch = "V-notchE3.sci"
                        Case Else
                              getModelNameV_Notch = "invalid notch width"
                  End Select
            Case 0.25 To (2# + tol)    'nh
                  Select Case nw
                        Case 0.03 To nwmax
                              getModelNameV_Notch = "V-notchD1.sci"
                        Case Else
                              getModelNameV_Notch = "invalid notch width"
                  End Select
            Case Else    'nh
                  getModelNameV_Notch = "invalid notch height"
         End Select
      Case Else    'a
         getModelNameV_Notch = "invalid value of a"
   End Select

Case 0.2 To (0.3 - tol)
   Select Case a 'crack length
      Case 0.005 To (0.05 - tol)
         Select Case nh   'notch height
            Case 0.22 To (0.3 - tol)
                  Select Case nw   'notch width
                        Case 0.03 To (0.1 - tol)
                              getModelNameV_Notch = "V-notchC4.sci"
                        Case 0.1 To nwmax
                              getModelNameV_Notch = "V-notchB1.sci"
                        Case Else
                              getModelNameV_Notch = "invalid notch width"
                  End Select
            Case 0.3 To (0.5 - tol) 'nh
                  Select Case nw   'notch width
                        Case 0.03 To (0.1 - tol)
                              getModelNameV_Notch = "V-notchA5.sci"
                        Case 0.1 To nwmax
                              getModelNameV_Notch = "V-notchB1.sci"
                        Case Else
                              getModelNameV_Notch = "invalid notch width"
                  End Select
            Case 0.5 To (1# - tol)   'nh
```

```
                    Select Case nw   'notch width
                        Case 0.03 To (0.1 - tol)
                            getModelNameV_Notch = "V-notchA7.sci"
                        Case 0.1 To (0.4 - tol)
                            getModelNameV_Notch = "V-notchC4.sci"
                        Case 0.4 To nwmax
                            getModelNameV_Notch = "V-notchB2.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case 1# To (2# + tol) 'nh
                    Select Case nw   'notch width
                        Case 0.03 To (0.4 - tol)
                            getModelNameV_Notch = "V-notchA7.sci"
                        Case 0.4 To nwmax
                            getModelNameV_Notch = "V-notchC4.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case Else     'nh
                    getModelNameV_Notch = "invalid notch height"
            End Select
        Case 0.05 To amax
            Select Case nh
                Case (1.1 * r) To (2# + tol)
                    Select Case nw
                        Case 0.03 To nwmax
                            getModelNameV_Notch = "V-notchD1.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case Else    'nh
                    getModelNameV_Notch = "invalid notch height"
            End Select
        Case Else    'a
            getModelNameV_Notch = "invalid value for a"
    End Select

 Case 0.3 To (0.4 - tol)
    Select Case a  'crack length
        Case 0.005 To (0.05 - tol)
            Select Case nh  'notch height
                Case 0.33 To (0.4 - tol)
                    Select Case nw  'notch width
                        Case 0.03 To (0.1 - tol)
                            getModelNameV_Notch = "V-notchC4.sci"
                        Case 0.1 To nwmax
                            getModelNameV_Notch = "V-notchB1.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case 0.4 To (1# - tol)  'nh
                    Select Case nw   'notch width
                        Case 0.03 To (0.1 - tol)
                            getModelNameV_Notch = "V-notchA7.sci"
```

```
                                Case 0.1 To (0.4 - tol)
                                    getModelNameV_Notch = "V-notchC4.sci"
                                Case 0.4 To nwmax
                                    getModelNameV_Notch = "V-notchB2.sci"
                                Case Else
                                    getModelNameV_Notch = "invalid notch width"
                            End Select
                      Case 1# To (2# + tol)   'nh
                            Select Case nw   'notch width
                                Case 0.03 To (0.4 - tol)
                                    getModelNameV_Notch = "V-notchA7.sci"
                                Case 0.4 To nwmax
                                    getModelNameV_Notch = "V-notchB2.sci"
                                Case Else
                                    getModelNameV_Notch = "invalid notch width"
                            End Select
                      Case Else     'nh
                            getModelNameV_Notch = "invalid notch height"
                  End Select
            Case 0.05 To amax
                  Select Case nh
                      Case (1.1 * r) To (2# + tol)
                            Select Case nw
                                Case 0.03 To nwmax
                                    getModelNameV_Notch = "V-notchD1.sci"
                                Case Else
                                    getModelNameV_Notch = "invalid notch width"
                            End Select
                      Case Else    'nh
                            getModelNameV_Notch = "invalid notch height"
                  End Select
            Case Else     'a
                  getModelNameV_Notch = "invalid value for a"
          End Select

    Case 0.4 To (0.5 - tol)
        Select Case a   'crack length
            Case 0.005 To (0.05 - tol)
                  Select Case nh   'notch height
                      Case 0.44 To (0.5 - tol)
                            Select Case nw   'notch width
                                Case 0.03 To (0.1 - tol)
                                    getModelNameV_Notch = "V-notchC4.sci"
                                Case 0.1 To nwmax
                                    getModelNameV_Notch = "V-notchB1.sci"
                                Case Else
                                    getModelNameV_Notch = "invalid notch width"
                            End Select
                      Case 0.5 To (1# - tol)   'nh
                            Select Case nw   'notch width
                                Case 0.03 To (0.1 - tol)
                                    getModelNameV_Notch = "V-notchA7.sci"
                                Case 0.1 To nwmax
                                    getModelNameV_Notch = "V-notchB1.sci"
                                Case Else
```

```
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 1# To (2# + tol)  'nh
                        Select Case nw   'notch width
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchA7.sci"
                            Case 0.1 To (0.4 - tol)
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case 0.4 To nwmax
                                getModelNameV_Notch = "V-notchB2.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case Else     'nh
                        getModelNameV_Notch = "invalid notch height"
                End Select
            Case 0.05 To amax
                Select Case nh
                    Case (1.1 * r) To (2# + tol)
                        Select Case nw
                            Case 0.03 To (0.4 - tol)
                                getModelNameV_Notch = "V-notchD3.sci"
                            Case 0.4 To nwmax
                                getModelNameV_Notch = "V-notchD2.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case Else    'nh
                        getModelNameV_Notch = "invalid notch height"
                End Select
            Case Else    'a
                getModelNameV_Notch = "invalid value for a"
        End Select

        Case 0.5 To (0.6 - tol)
        Select Case a   'crack length
            Case 0.005 To (0.05 - tol)
                Select Case nh   'notch height
                    Case 0.55 To (0.6 - tol)
                        Select Case nw   'notch width
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchB2.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 0.6 To (1# - tol)   'nh
                        Select Case nw   'notch width
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchA7.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchB2.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
```

```
                End Select
            Case 1# To (2# + tol)   'nh
                Select Case nw   'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchA7.sci"
                    Case 0.1 To (0.4 - tol)
                        getModelNameV_Notch = "V-notchC4.sci"
                    Case 0.4 To nwmax
                        getModelNameV_Notch = "V-notchB1.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else     'nh
                getModelNameV_Notch = "invalid notch height"
        End Select
    Case 0.05 To amax
        Select Case nh
            Case (1.1 * r) To (2# + tol)
                Select Case nw
                    Case 0.03 To (0.4 - tol)
                        getModelNameV_Notch = "V-notchD3.sci"
                    Case 0.4 To nwmax
                        getModelNameV_Notch = "V-notchD2.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else     'nh
                getModelNameV_Notch = "invalid notch height"
        End Select
    Case Else     'a
        getModelNameV_Notch = "invalid value for a"
End Select

Case 0.6 To (0.7 - tol)
Select Case a   'crack length
    Case 0.005 To (0.05 - tol)
        Select Case nh   'notch height
            Case 0.66 To (0.7 - tol)
                Select Case nw   'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchC4.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchB3.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case 0.7 To (1# - tol)   'nh
                Select Case nw   'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchA7.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchB3.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
```

```
                Case 1# To (2# + tol)   'nh
                    Select Case nw   'notch width
                        Case 0.03 To (0.1 - tol)
                            getModelNameV_Notch = "V-notchA7.sci"
                        Case 0.1 To nwmax
                            getModelNameV_Notch = "V-notchC5.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case Else    'nh
                    getModelNameV_Notch = "invalid notch height"
            End Select
        Case 0.05 To amax
            Select Case nh
                Case (1.1 * r) To (2# + tol)
                    Select Case nw
                        Case 0.03 To (0.4 - tol)
                            getModelNameV_Notch = "V-notchD3.sci"
                        Case 0.4 To nwmax
                            getModelNameV_Notch = "V-notchD2.sci"
                        Case Else
                            getModelNameV_Notch = "invalid notch width"
                    End Select
                Case Else   'nh
                    getModelNameV_Notch = "invalid notch height"
            End Select
        Case Else    'a
            getModelNameV_Notch = "invalid value for a"
    End Select

Case 0.7 To (0.8 - tol)
Select Case a   'crack length
    Case 0.005 To (0.05 - tol)
        Select Case nh   'notch height
            Case (1.1 * r) To (1# - tol)
                Select Case nw   'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchC4.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchB3.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case 1# To (2# + tol)   'nh
                Select Case nw   'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchA7.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchC5.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else    'nh
                getModelNameV_Notch = "invalid notch height"
        End Select
```

```
    Case 0.05 To amax
        Select Case nh
            Case (1.1 * r) To (2# + tol)
                Select Case nw
                    Case 0.03 To (0.4 - tol)
                        getModelNameV_Notch = "V-notchD3.sci"
                    Case 0.4 To nwmax
                        getModelNameV_Notch = "V-notchD2.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else   'nh
                getModelNameV_Notch = "invalid notch height"
        End Select
    Case Else    'a
        getModelNameV_Notch = "invalid value for a"
End Select

Case 0.8 To (0.9 - tol)
Select Case a  'crack length
    Case 0.005 To (0.05 - tol)
        Select Case nh  'notch height
            Case (1.1 * r) To (1# - tol)
                Select Case nw  'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchC4.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchB3.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case 1# To (2# + tol)  'nh
                Select Case nw  'notch width
                    Case 0.03 To (0.1 - tol)
                        getModelNameV_Notch = "V-notchA7.sci"
                    Case 0.1 To nwmax
                        getModelNameV_Notch = "V-notchB3.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else     'nh
                getModelNameV_Notch = "invalid notch height"
        End Select
    Case 0.05 To amax
        Select Case nh
            Case (1.1 * r) To (2# + tol)
                Select Case nw
                    Case 0.03 To (0.4 - tol)
                        getModelNameV_Notch = "V-notchD3.sci"
                    Case 0.4 To nwmax
                        getModelNameV_Notch = "V-notchD2.sci"
                    Case Else
                        getModelNameV_Notch = "invalid notch width"
                End Select
            Case Else    'nh
```

```
                        getModelNameV_Notch = "invalid notch height"
                End Select
            Case Else    'a
                getModelNameV_Notch = "invalid value for a"
        End Select

        Case 0.9 To (1# - tol)
        Select Case a   'crack length
            Case 0.005 To (0.05 - tol)
                Select Case nh  'notch height
                    Case 0.99 To (1# - tol)
                        Select Case nw  'notch width
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchC4.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchB3.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case 1# To (2# + tol)  'nh
                        Select Case nw  'notch width
                            Case 0.03 To (0.1 - tol)
                                getModelNameV_Notch = "V-notchA7.sci"
                            Case 0.1 To nwmax
                                getModelNameV_Notch = "V-notchB3.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case Else     'nh
                        getModelNameV_Notch = "invalid notch height"
                End Select
            Case 0.05 To amax
                Select Case nh
                    Case (1.1 * r) To (2# + tol)
                        Select Case nw
                            Case 0.03 To (0.4 - tol)
                                getModelNameV_Notch = "V-notchD3.sci"
                            Case 0.4 To nwmax
                                getModelNameV_Notch = "V-notchD2.sci"
                            Case Else
                                getModelNameV_Notch = "invalid notch width"
                        End Select
                    Case Else    'nh
                        getModelNameV_Notch = "invalid notch height"
                End Select
            Case Else    'a
                getModelNameV_Notch = "invalid value for a"
        End Select
    Case Else   'r
        getModelNameV_Notch = "invalid value of notch radius"
End Select
End Function
```

6.6.7 Sample Results from FE Model(s)

6.6.7.1 Sample results for model *V-notchC4.sci.* StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCeck Parameter Name	AFGROW Parameter value	StressCheck Parameter Value	AFGROW Parameter value
a	c	0.005 in.	0.005 in.
dim	N/A	0.15	N/A
h	N/A	4 in.	N/A
nh	nh	1.0 in.	1.0 in.
nw	nw	0.40 in.	0.40 in.
rd	R	0.10 in.	0.10 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 1.240 ksi-in$^{1/2}$ and β = 9.894. The StressCheck computed Limit K1 is 1.240 ksi-in$^{1/2}$.

Limit K1 for use in AFGROW = 1.819 / sqrt(modelScale) = 1.819 / sqrt(10) = 0.5752.

6.6.7.2 Sample results for model *notchD1.sci*. StressCheck version V6.3.1f (Beta) was used. Integration radius = 0.075*a.

StressCeck Parameter Name	AFGROW Parameter value	StressCheck Parameter Value	AFGROW Parameter value
a	c	0.50 in.	0.50 in.
dim	N/A	0.15	N/A
h	N/A	4 in.	N/A
nh	nh	1.0 in.	1.0 in.
nw	nw	0.80 in.	0.80 in.
rd	R	0.10 in.	0.10 in.
St	SMF	1 ksi	1 ksi
th	T	0.1 in.	0.1 in.
w	W	1 in.	1 in.

Using the values as shown in the table above, the StressCheck computed Mode I Stress Intensity Factor K1 at p-level 8 is 0.3003 ksi-in$^{1/2}$ and β = 0.240. The StressCheck computed Limit K1 is 0.3003 ksi-in$^{1/2}$.

6.6.7.3 Sample results for various models and notch configurations. StressCheck version V6.3.1f (Beta) was used. For all input h = 4, W = 1, and St = 1. Integration radius = 0.075*a.

*.sci Parameter Name	Crack Length a	Notch Height nh	Notch Width nw	Notch Radius R	Stress Intensity K1	β
V-notchC4.sci	0.005	0.50	0.10	0.10	3.447e-01	2.750
V-notchC4.sci	0.005	1.00	0.40	0.10	1.240e+00	9.894
V-notchB1.sci	0.005	0.50	0.10	0.40	2.877e-01	2.296
V-notchB1.sci	0.005	1.00	0.80	0.40	9.842e+00	78.528
V-notchA7.sci	0.005	1.00	0.03	0.80	1.737e-01	1.386
V-notchA7.sci	0.005	2.00	0.10	0.80	2.356e-01	1.880
V-notchD2.sci	0.25	1.00	0.50	0.40	1.297e+01	14.635
V-notchD2.sci	0.05	2.00	0.40	0.60	2.390e+00	6.030
V-notchD3.sci	0.50	1.00	0.03	0.10	4.019e+00	3.207
V-notchD3.sci	0.25	2.00	0.40	0.20	7.099e+00	8.010

**6.7   Model Family Documentation: Through Crack at a Plate Nut**

6.7.1 Introduction

The goal of this family of StressCheck models is to provide models for use with the "Nut Plate Through Crack in a Satellite Hole" plug-in module (external-K component) for AFGROW. The models are defined in accordance with the geometric limitations specified herein. All finite element models are two-dimensional, auto-meshed models subjected to tension, bearing and shear stress loads.

6.7.2 Software Version

The StressCheck models use StressCheck Version 7.0.1f.

6.7.3 Sketch of Model Parameters (AFGROW definitions)

This section provides a sketch of a typical AFGROW user interface for these models. Included in the sketch are the parameter names exposed to the AFGROW interface.
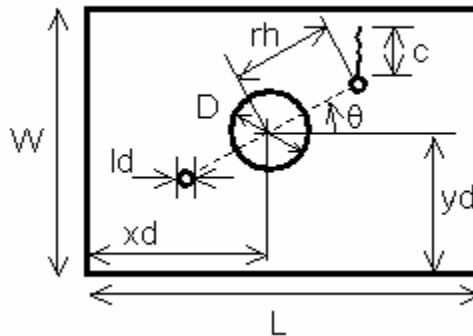
6.7.4 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the AFGROW interface and family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

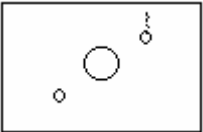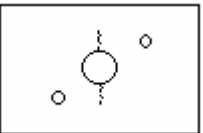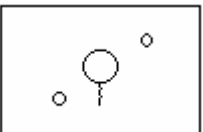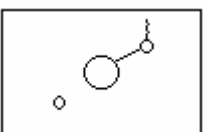**6.7.4.1 Parameters in the User Interface (Length dimensions in inches)**
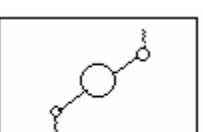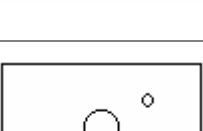
AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
L	l	Plate X-length	no limit	no limit
W	w	Plate Y-width	no limit	no limit
D	d	Main Hole Diameter	geometric	geometric
xd	xd	Main Hole X-Center	geometric	geometric
yd	yd	Main Hole Y-Center	geometric	geometric
ld	ld	Satellite Hole Diameter	geometric	geometric
theta	theta	Satellite Hole Angle (degrees)	geometric	geometric
rh	rh	Satellite-to-Main Hole Center Distance	geometric	geometric
th	th	Plate Thickness	no limit	no limit
c		Initial Crack Length	0.005	geometric
Modulus	modulus	Young's modulus (ksi)	no limit	no limit
pby1		X direction bypass force (lb)	no limit	no limit
pby2		shear bypass force on top face (lb)	no limit	no limit
paxsh1		force on positive X face, reacted in shear on bottom face (lb)	no limit	no limit
paxsh2		force on positive X face, reacted in shear on top face (lb)	no limit	no limit
paxsh3		force on negative X face, reacted in shear on bottom face (lb)	no limit	no limit
paxsh4		force on negative X face, reacted in shear on top face (lb)	no limit	no limit
pbrax1		bearing force in positive X direction, reacted	no limit	no limit

| AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
		axially on left face (lb)		
pbrax2		bearing force in negative X direction, reacted axially on right face (lb)	no limit	no limit
pbrax3		bearing force in positive Y direction, reacted axially on bottom face (lb)	no limit	no limit
pbrax4		bearing force in negative Y direction, reacted axially on top face (lb)	no limit	no limit
pbrsh1		bearing force in positive X direction, reacted in shear on bottom face (lb)	no limit	no limit
pbrsh2		bearing force in positive X direction, reacted in shear on top face (lb)	no limit	no limit
pbrsh3		bearing force in positive Y direction, reacted in shear on left face (lb)	no limit	no limit
pbrsh4		bearing force in positive Y direction, reacted in shear on right face (lb)	no limit	no limit

6.7.4.2 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Expression	Detail Description
brx	x bearing force		Bearing force in X direction.
bry	y bearing force		Bearing force in Y direction.
bx	bottom tangent force		Tangential force on bottom face.
by	bottom normal force		Normal force on bottom face.
lx	left normal force		Normal force on left face.
ly	left tangent force		Tangential force on left face.
modulus	plate modulus		Young's modulus.
rx	right normal force		Normal force on right face.
ry	right tangent force		Tangential force on right face.
sbx	bottom tangent stress	bx/(th*l)	Calculated stress on bottom face.
sby	bottom normal stress	by/(th*l)	Calculated stress on bottom face.
slx	left normal stress	lx/(th*w)	Calculated stress on left face.
sly	left tangent stress	ly/(th*w)	Calculated stress on left face.
srx	right normal stress	rx/(th*w)	Calculated stress on right face.
sry	right tangent stress	ry/(th*w)	Calculated stress on right face.
stx	top tangent stress	-tx/(th*l)	Calculated stress on top face.
sty	top normal stress	ty/(th*l)	Calculated stress on top face.
tx	top tangent force		Tangential force on top face.
ty	top normal force		Normal force on top face.

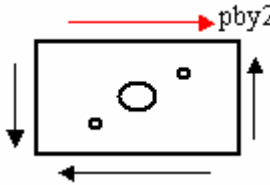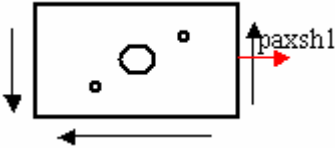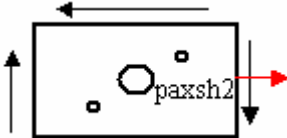## 6.7.5 File Names, Dates, Brief Description and Diagrams
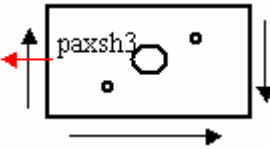
*.sci File Name	File Date (mm.dd.yyyy)	Brief Description	Diagram
final_nutplate_single	12/16/2004	Nut plate with a single crack originating from a satellite hole.	
final_main_double	12/16/2004	Nut plate with two cracks originating from the main hole.	
final_main_single	12/16/2004	Nut plate with a single crack originating from the main hole.	
final_single_slot	12/16/2004	Nut plate with a ligament broken between the main hole and a satellite hole, and a single crack originating from the satellite hole.	
final_double_slot	12/16/2004	Nut plate with both ligaments broken between the main hole and the satellite holes, and a single crack originating from each satellite hole.	
final_crack_to_main	12/16/2004	Nut plate with a single crack originating from a satellite hole, growing towards the main hole.	
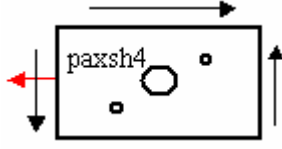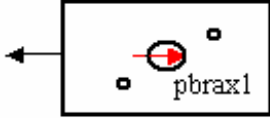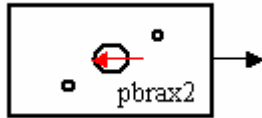
6.7.6 Load Conditions
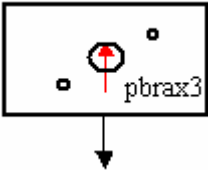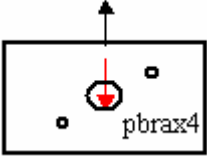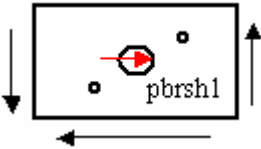
There are 14 load inputs in the AFGROW userface. Any combination of load conditions (bypass, bearing, shear) can be specified by the user. The user may also specify the force reaction face. Equilibrium reaction forces are calculated in AFGROW and then input into StressCheck, which then calculates the stresses and applies them to each face. A general schematic using the StressCheck parameter names for force loads is shown below.
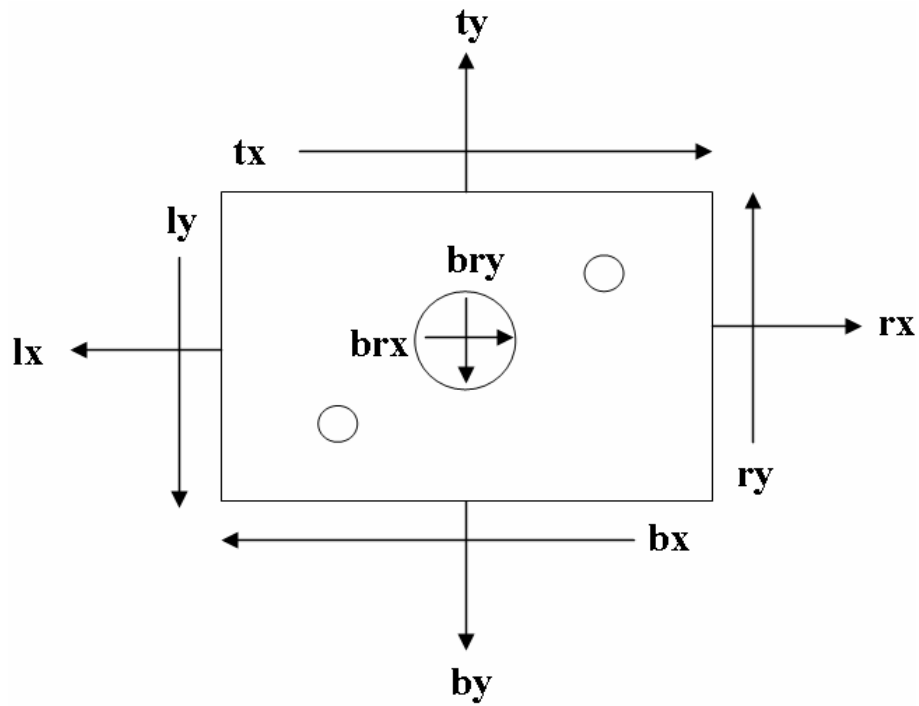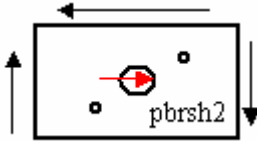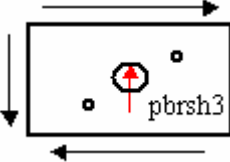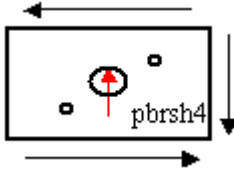
Each loading condition, including equilibrium calculations and diagrams, is shown in the table below. Note that since multiple loads can be input in any combination, the reaction/equilibrium calculations are cumulated values. Also note the actual input force is the AFGROW stress multiplication factor, SMF multiplied by the user input force.

Load Case	Load Input	Reaction/Equilibrium Calculations
	X direction bypass force, pby1	$rx = rx + pby1$ $lx = lx + rx$
	shear bypass force on top face, pby2	$tx = tx + pby2$ $bx = bx - pby2$ $ly = ly + pby2*w/l$ $ry = ry + pby2*w/l$
	force on positive X face, reacted in shear on bottom face, paxsh1	$rx = rx + paxsh1$ $bx = bx - paxsh1$ $ly = ly + paxsh1*(w/2)/l$ $ry = ry + paxsh1*(w/2)/1$
	force on positive X face, reacted in shear on top face, paxsh2	$rx = rx + paxsh2$ $tx = tx - paxsh2$ $ly = ly - paxsh2*(w/2)/l$ $ry = ry - paxsh2*(w/2)/1$
	force on negative X face, reacted in shear on bottom face, paxsh3	$lx = lx + paxsh3$ $bx = bx + paxsh3$ $ly = ly - paxsh3*(w/2)/l$ $ry = ry - paxsh3*(w/2)/l$

Load Case	Load Input	Reaction/Equilibrium Calculations
	force on negative X face, reacted in shear on top face, paxsh4	lx = lx + paxsh4 tx = tx + paxsh4 ly = ly + paxsh4*(w/2)/l ry = ry + paxsh4*(w/2)/l
	bearing force in positive X direction, reacted axially on left face, pbrax1	brx = brx + pbrax1 lx = lx + pbrax1 Case 1: yd < w/2 m = pbrax1*(w/2 – yd) ly = ly – m/l ry = ry – m/l Case 2: yd > w/2 m = pbrax1*(yd – w/2) ly = ly + m/l ry = ry + m/l
	bearing force in negative X direction, reacted axially on right face, pbrax2	brx = brx – pbrax2 rx = rx + pbrax2 Case 1: yd < w/2 m = pbrax2*(w/2 – yd) ly = ly + m/l ry = ry + m/l Case 2: yd > w/2 m = pbrax2*(yd – w/2) ly = ly – m/l ry = ry – m/l

Load Case	Load Input	Reaction/Equilibrium Calculations

	bearing force in positive Y direction, reacted axially on bottom face, pbrax3	bry = bry + pbrax3
		
		by = by + pbrax3
		
		Case 1: xd < l/2
		
		   m = pbrax3*(l/2 – xd)
		
		   ly = ly + m/l
		
		   ry = ry + m/l
		
		Case 2: xd > l/2
		
		   m = pbrax3*(xd – l/2)
		
		ly = ly – m/l
		
		ry = ry – m/l

	bearing force in negative Y direction, reacted axially on top face, pbrax4	bry = bry – pbrax4
		
		ty = ty + pbrax4
		
		Case 1: xd < l/2
		
		   m = pbrax4*(l/2 – xd)
		
		   ly = ly – m/l
		
		   ry = ry – m/l
		
		Case 2: xd > l/2
		
		   m = pbrax4*(xd – l/2)
		
		ly = ly + m/l
		
		ry = ry + m/l

	bearing force in positive X direction, reacted in shear on bottom face, pbrsh1	brx = brx + pbrsh1 bx = bx – pbrsh1 ly = ly + pbrsh1*yd/l ry = ry + pbrsh1*yd/l

Load Case	Load Input	Reaction/Equilibrium Calculations
	bearing force in positive X direction, reacted in shear on top face, pbrsh2	brx = brx + pbrsh2 tx = tx – pbrsh2 ly = ly – (pbrsh2*(w – yd))/l ry = ry – (pbrsh2*(w – yd))/l
	bearing force in positive Y direction, reacted in shear on left face, pbrsh3	bry = bry + pbrsh3 ly = ly + pbrsh3 tx = tx + pbrsh3*xd/w bx = bx – pbrsh3*xd/w
	bearing force in positive Y direction, reacted in shear on right face, pbrsh4	bry = bry + pbrsh4 ry = ry – pbrsh4 tx = tx – (pbrsh4*(1 – d))/w bx = bx +(pbrsh4*(1–xd))/w

6.7.7 Model Constraints

The nut plate is constrained in the X and Y direction with a two-point rigid body constraint.

6.8   Model Family Documentation: Part-Through Crack at a Lug

6.8.1 Introduction

The goal of this family of StressCheck models is to provide a series of tabulated tables of Stress Intensity Factors for a Single Corner Crack in a Lug for AFGROW. All finite element models are three-dimensional models subject to bearing stress.

6.8.2 Sketch of Model Parameters (AFGROW definitions)

This section provides a sketch of the AFGROW user interface. Included in the sketch are the parameter names exposed to the AFGROW interface.
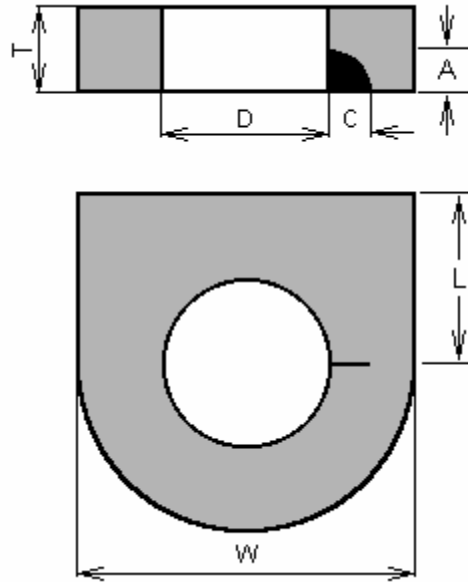
6.8.3 Model Scale

Scaling the finite-element model is often necessary to avoid inherent numerical difficulties with very small numbers. **The model scale for this family of models is 10.0**. The AFGROW geometric parameter values and limits are multiplied by the model scale before they are used by StressCheck, and the resulting StressCheck Stress Intensity Factors are divided by the square root of the model scale, then used to calculate tabulated beta factors for use in AFGROW.

6.8.4 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the family of StressCheck input (*.sci) files. The first section lists parameter names related to the AFGROW interface; the second section defines parameters used only by the finite-element models.

6.8.4.1 Parameters in the User Interface (Length dimensions in inches)

AFGROW Parameter	*.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
a	a	crack bore length	0.01*T	0.90*T
c	c	crack surface length	0.005	1.80
D	Dh	hole diameter	0.50*T	4.0*T
T	th	thickness	No limit	No limit
W	Ro	lug outer radius	1.5*D	2.5*D

6.8.4.2 Parameters Specific to the Finite-Element Models

*.sci Parameter Name	*.sci Parameter Description	Detail Description
dim	dimension	Model construction parameter. The size of the local mesh around the crack tip increases and decreases with this parameter.
Fpin	pin force	Force parameter. The applied load is 1*D*T for a bearing stress of 1 psi.
L	length	Model length as measured from the center of the lug hole. Fixed at L=W.

6.8.5 Parameter Bounds

Parameter boundaries are shown below. Note that not all boundary combinations are valid within geometric and model constraints, e.g., for parameter a/T = 0.01, the upper bound for a/c = 2.0 in that the minimum crack length c must be ≤ 0.005.

Parameter	Lower Bound	Upper Bound
a/T	0.01	0.90
a/c	0.50	4.0
D/T	0.50	4.0
W/D	1.5	2.5

6.8.6 File Names, Dates, and Brief Description

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
lug1C.sci lug1C4.sci	11/2/2004	Small a/t ratio
lug2C.sci	11/2/2004	Small to medium a/t ratio
lug3C.sci lug3C1.sci	11/2/2004	Medium to large a/t ratio
lug4C.sci lug4C1.sci lug4C2.sci	11/2/2004	Large a/t ratio

Valid crack length and notch radius parameter ranges for each file:

Crack Length/ Thickness (a/t)	Crack Ratio (a/c)	Diameter/ Thickness (D/T)	Width/ Diameter (W/D)	*.sci File Name
$0.005 \leq a/t < 0.01$	$0.50 \leq a/c \leq 2.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug1C4.sci
$0.01 \leq a/t < 0.05$	$0.50 \leq a/c \leq 4.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug1C.sci lug1C4.sci
$0.05 \leq a/t < 0.10$	$0.50 \leq a/c \leq 4.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug1C.sci lug2C.sci lug3C.sci
$0.10 \leq a/t < 0.20$	$0.50 \leq a/c \leq 4.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug2C.sci lug3C.sci lug3C1.sci
$0.20 \leq a/t < 0.80$	$0.50 \leq a/c \leq 4.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug3C.sci lug3C1.sci
$0.80 \leq a/t < 0.90$	$0.50 \leq a/c \leq 4.0$	$0.5 \leq D/T \leq 4.0$	$1.5 \leq W/D \leq 2.5$	lug4C.sci lug4C1.sci lug4C2.sci

6.8.7 Sample Snapshots of Finite Element Model(s)

Model lug3C.sci is shown below depicting the bearing stress, symmetry and nodal constraints.

6.8.8 Sample data using FE Models

The look-up tables represent a comprehensive array of parameter combinations within the defined parameter limits. The K_{1a} and K_{1c} are defined as follows:

$$K_{1a} = \beta_a \sqrt{\pi c}$$

$$K_{1c} = \beta_c \sqrt{\pi c}$$

where K_{1a} is the local maxima of the Stress Intensity Factors along the crack length near the lug bore, and K_{1c} is the local maxima of the Stress Intensity Factors along the crack length near the lug surface.
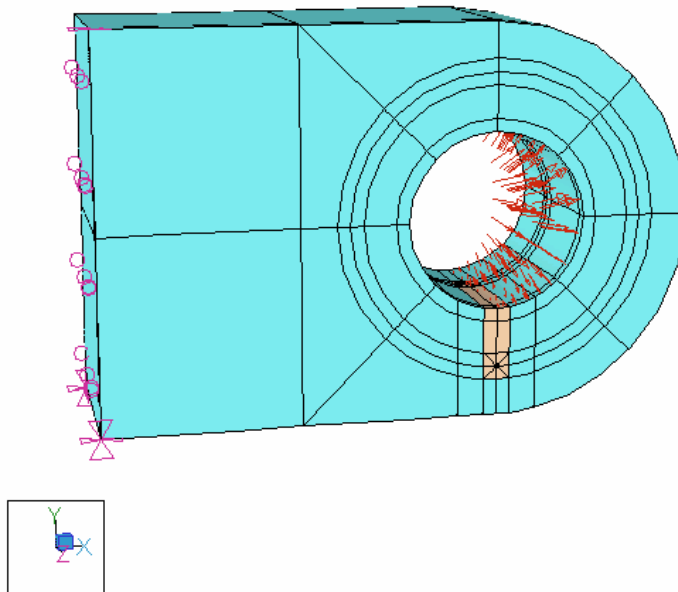
Sample results for various models and configurations are shown below. StressCheck version V6.3.2b was used. For those combinations not directly located in the tables, the K_{1a} and K_{1c} values are to be interpolated.

*.sci File Name	A/T	A/C	D/T	W/D	K_{1a}	K_{1c}	β_a	β_c	Conv. (%)
lug1C4.sci	0.005	0.5	0.5	1.5	0.588	0.393	1.483	0.702	0.865
lug1C4.sci	0.005	0.5	0.5	2.5	0.214	0.147	0.54	0.262	0.332
lug1C4.sci	0.005	0.5	4	1.5	0.666	0.499	1.681	0.89	0.266
lug1C4.sci	0.005	0.5	4	2.5	0.25	0.186	0.63	0.333	0.59
lug1C4.sci	0.005	1	4	1.5	0.528	0.534	1.332	1.348	0.274
lug1C.sci	0.01	0.5	0.5	1.5	0.795	0.473	1.419	0.597	0.163
lug1C.sci	0.01	0.5	0.5	2.5	0.293	0.184	0.523	0.232	0.722
lug1C4.sci	0.01	0.5	4	1.5	0.946	0.699	1.688	0.882	0.242
lug1C4.sci	0.01	0.5	4	2.5	0.356	0.264	0.635	0.333	0.513
lug1C4.sci	0.01	2	4	1.5	0.521	0.677	0.929	1.709	0.501
lug1C4.sci	0.01	2	4	2.5	0.194	0.251	0.345	0.634	0.461
lug2C.sci	0.05	0.5	0.5	2.5	0.561	0.249	0.448	0.141	0.116
lug1C.sci	0.05	0.5	4	1.5	2.032	1.354	1.621	0.764	0.756
lug1C.sci	0.05	0.5	4	2.5	0.767	0.53	0.612	0.299	0.686
lug2C.sci	0.05	4	0.5	1.5	0.705	0.969	0.563	1.547	0.315
lug2C.sci	0.05	4	0.5	2.5	0.259	0.355	0.207	0.567	0.666
lug3C.sci	0.1	0.5	0.5	2.5	0.689	0.249	0.389	0.099	0.557
lug2C.sci	0.1	0.5	4	1.5	2.763	1.646	1.559	0.657	0.407
lug2C.sci	0.1	0.5	4	2.5	1.057	0.684	0.596	0.273	0.606
lug2C.sci	0.1	4	1	1.5	1.015	1.392	0.573	1.571	0.34
lug2C.sci	0.1	4	1	2.5	0.375	0.521	0.212	0.588	0.54
lug3C.sci	0.2	0.5	4	1.5	3.725	1.784	1.486	0.503	0.33
lug3C.sci	0.2	0.5	4	2.5	1.425	0.827	0.568	0.233	0.852
lug3C.sci	0.2	4	0.5	1.5	1.467	1.658	0.585	1.323	0.353
lug3C.sci	0.2	4	0.5	2.5	0.507	0.572	0.202	0.457	0.417
lug3C1.sci	0.2	4	2	1.5	1.466	2.05	0.585	1.635	0.475
lug3C1.sci	0.2	4	2	2.5	0.55	0.792	0.219	0.632	0.776
lug3C.sci	0.4	0.5	4	1.5	5.426	2.024	1.531	0.404	0.069
lug3C.sci	0.4	0.5	4	2.5	1.947	1.038	0.549	0.207	0.163
lug3C.sci	0.4	4	0.5	2.5	0.654	0.696	0.184	0.392	0.085
lug3C1.sci	0.4	4	4	1.5	2.097	3.061	0.592	1.727	0.155
lug3C1.sci	0.4	4	4	2.5	0.785	1.177	0.221	0.664	0.454
lug3C.sci	0.6	0.5	4	2.5	2.391	1.333	0.551	0.217	0.076
lug3C.sci	0.6	4	0.5	2.5	0.74	0.805	0.171	0.371	0.08
lug3C1.sci	0.6	4	4	1.5	2.57	3.64	0.592	1.677	0.081
lug3C1.sci	0.6	4	4	2.5	0.95	1.391	0.219	0.641	0.148
lug3C1.sci	0.7	0.5	4	2.5	2.581	1.531	0.55	0.231	0.057
lug4C.sci	0.7	4	0.5	2.5	0.758	0.857	0.162	0.366	0.151
lug3C1.sci	0.7	4	4	1.5	2.785	3.906	0.594	1.666	0.072
lug3C1.sci	0.7	4	4	2.5	1.02	1.481	0.218	0.632	0.111
lug4C.sci	0.8	0.5	4	2.5	2.744	1.796	0.547	0.253	0.185
lug4C.sci	0.8	4	0.5	2.5	0.789	0.914	0.157	0.364	0.135
lug4C.sci	0.9	0.5	4	2.5	2.914	2.05	0.548	0.273	0.13
lug4C.sci	0.9	4	0.5	2.5	0.832	0.97	0.157	0.365	0.129

6.9   Model Family Documentation: Multi-site Damage

6.9.1 Introduction

The goal of this family of StressCheck models is to provide models for use with the "Multi-Site Damage (MSD) in a Fixed Geometry Specimen" plug-in module (external-K component) for AFGROW. The model is defined in accordance with the geometric limitations specified herein. The finite element model is a two-dimensional, auto-meshed models subjected to tensile stress.

6.9.2 Software Version

The StressCheck models use StressCheck Version 7.0.1f.

6.9.3 Sketch of Model Parameters (AFGROW definitions)

This section provides a sketch of a typical AFGROW user interface for these models. Included in the sketch are the parameter names exposed to the AFGROW interface. Note the parameters W, L, S, ED, and D are fixed dimensions.
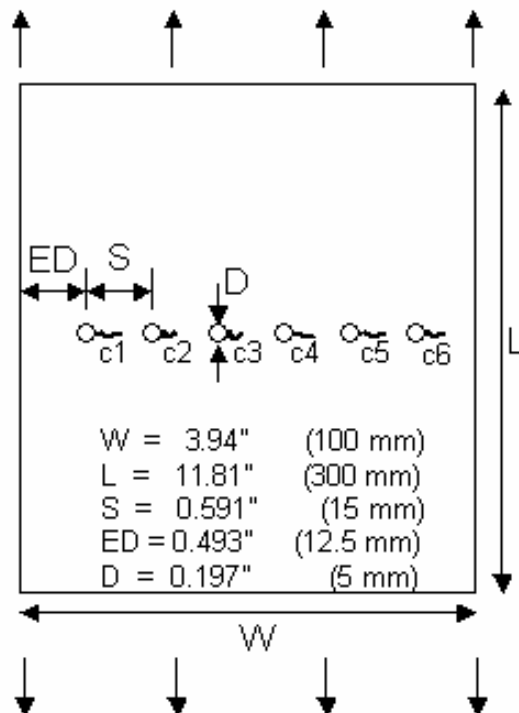
6.9.4 Parameter Names, Definitions, and Limits

This section contains the parameter names as defined in the AFGROW interface and StressCheck input (msd6.sci) file.

Parameters in the User Interface (Crack Length dimensions in inches)

AFGROW Parameter	msd6.sci Parameter Name	*.sci Parameter Description	Minimum AFGROW Limit	Maximum AFGROW Limit
c1	a6	crack 1 length	0.005	0.364
c2	a5	crack 2 length	0.005	0.364
c3	a4	crack 3 length	0.005	0.364
c4	a3	crack 4 length	0.005	0.364
c5	a2	crack 5 length	0.005	0.364
c6	a1	crack 6 length	0.005	0.364

6.9.5 File Names, Dates, Brief Description and Diagrams

*.sci File Name	File Date (mm.dd.yyyy)	Brief Description
msd6a.sci	01/03/2005	Multi-Site Damage in a Fixed Geometry Plate